Advanced turbulence modeling for industrial flows

Raymond Michael Jones

Louisiana State University and Agricultural and Mechanical College

Follow this and additional works at: https://digitalcommons.lsu.edu/gradschool_dissertations

Part of the Mechanical Engineering Commons

Recommended Citation

https://digitalcommons.lsu.edu/gradschool_dissertations/2515

This Dissertation is brought to you for free and open access by the Graduate School at LSU Digital Commons. It has been accepted for inclusion in LSU Doctoral Dissertations by an authorized graduate school editor of LSU Digital Commons. For more information, please contact gradetd@lsu.edu.
ADVANCED TURBULENCE MODELING FOR INDUSTRIAL FLOWS

A Dissertation

Submitted to the Graduate Faculty of the
Louisiana State University and
Agricultural and Mechanical College
in partial fulfillment of the
requirements for the degree of
Doctor of Philosophy

in
The Department of Mechanical Engineering

by
Raymond Michael Jones
B.S. Louisiana State University, 1996
August 2003
Acknowledgments

I would like to thank Dr. Acharya for providing financial support over the past several years. I would like to thank Albert Harvey of the Dow Chemical Company for his extensive help with code development and technical guidance throughout this work. Without his help this work would not have been possible. I would also like to thank the Dow Chemical Company for their financial support throughout this work.
Table of Contents

Acknowledgments........................................................................................................................................................................ii

Abstract ................................................................................................................................................................................................ v

Chapter 1. Introduction.................................................................................................................................................... 1
  1.1 Reynolds Averaged Equations ........................................................................................................................ 2
  1.2 Outline of the Thesis ............................................................................................................................................. 6

Chapter 2. Conventional Turbulence Models .................................................................................................. 8
  2.1 Two-Equation Models........................................................................................................................................... 8
     2.1.1 $k - \varepsilon$ Model (LKE) .....................................................10
     2.1.2 Modified $k - \varepsilon$ Model (MKE) ...............................................12
     2.1.3 RNG $k - \varepsilon$ Model (RNG) ...............................................12
     2.1.4 $k - \omega$ Model (LKW),(HKW) .................................................13
     2.1.5 Modified $k - \omega$ Model (MKW) ..........................................15

Chapter 3. Stirred Tank.................................................................................................................................................. 16
  3.1 Mean Flow Equations......................................................................................................................................... 19
  3.2 Results and Discussion ...................................................................................................................................... 20
     3.2.1 Reactor Geometry and Operating Conditions .............................20
     3.2.2 Numerical Results .................................................................23
  3.3 Summary of Stirred Tank Results ............................................................................................................. 40

Chapter 4. Elliptic Relaxation and the Development of a New $v^2f$ Model ......................... 41
  4.1 Near Wall Velocity Scaling ........................................................................................................................ 41
  4.2 Elliptic Relaxation ............................................................................................................................................... 42
     4.2.1 The $v^2f - k\varepsilon$ Model ....................................................47
     4.2.2 The $v^2f - k\omega$ Model....................................................54

Chapter 5. Flow Solver.................................................................................................................................................. 63
  5.1 General Transport Equations ........................................................................................................................ 63
  5.2 Governing Equations........................................................................................................................................... 64
     5.2.1 Inviscid Fluxes .............................................................................65
     5.2.2 Viscous Fluxes ............................................................................66
     5.2.3 Preconditioning ...........................................................................67
     5.2.4 Numerical Method ........................................................................67

Chapter 6. Validation of the $v^2f - k\omega$ Model ............................................................................................. 70
  6.1 Two Dimensional Channel ............................................................................................................................. 70
  6.2 Backward Facing step ........................................................................................................................................ 73
  6.3 Cavity.............................................................................................................................................................................. 84
  6.4 Coaxial Jet ............................................................................................................................................................91
Abstract

This dissertation deals with the development of an improved two-equation turbulence model and its application to various flows. Six different conventional turbulence models were initially tested for predicting the flow inside of an unbaffled stirred tank reactor (STR), and the results are compared with experimental LDV data. Each of the models use low Reynolds number corrections. Results indicate that the radial velocity component in the impeller discharge region is overpredicted by each of the models. The tangential velocity component in the impeller discharge region is predicted well by the models, but is underpredicted near the shaft. The low Reynolds number $k - \varepsilon$ model is the only model which produces reasonable kinetic energy predictions in the impeller discharge region. The model predictions are generally unsatisfactory and produce varying results, which are largely attributed to the difference in the formulation of the low Reynolds number corrections for each model.

Based on these results a new model has been developed which eliminates the need for low Reynolds number corrections and has been demonstrated to produce improved results for various flows compared with the conventional models. The new model, called the $\overline{v^2}f - k\omega$ model, is developed based on the elliptic relaxation approach of Durbin [1] and is a variant of Durbin’s $\overline{v^2}f - k\varepsilon$ model. The new model is shown to be superior to all other models on a number of benchmark problems including the backstep, two-dimensional cavity, coaxial jet and jet in a crossflow. The new model is therefore proposed as a superior alternative from both computational effectiveness and accuracy viewpoints.
Chapter 1. Introduction

Computational fluid dynamics (CFD) in industry is typically performed using the Reynolds-averaged Navier-Stokes equations (RANS). With RANS, complex turbulent flows at high Reynolds numbers are represented by a turbulence model, and can be solved within an acceptable amount of time. At issue is the accuracy of RANS solutions. Other methods such as large eddy simulations (LES) and direct numerical simulations (DNS) can be used to obtain more accurate results but require extensive computational resources, even for complex turbulent flows at moderately low Reynolds numbers. Two-equation turbulence models are the most widely used models for predicting turbulent flows in industry. A wide range of two-equation models are available, each producing varying levels of success depending on the problem being solved. A common disadvantage of all turbulence models is that low-Reynolds number damping functions are needed to correctly predict near wall turbulence. Most two-equation models also fail to predict the law of the wall, and require these damping functions in order to produce accurate results. These damping functions are usually derived for simple wall bounded flows, such as flow in a channel, and are usually not suited for highly separated flows. This thesis shows the predictive capabilities of various two-equation models to predict a variety of different flows. Based on these results a new approach is introduced which helps to alleviate the need for damping functions. The goal of this thesis is to develop a new two-equation model which improves the predictive capabilities of conventional two-equation models without the need for damping functions. This new two-equation model is tested and
compared with other two-equation models for predicting various complex flows.

1.1 Reynolds Averaged Equations

The incompressible, constant-property equations for mass and momentum are

\[
\frac{\partial u_i}{\partial x_i} = 0 \quad (1.1)
\]

\[
\rho \frac{\partial u_i}{\partial t} + \rho u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial t_{ij}}{\partial x_j} \quad (1.2)
\]

\[
t_{ij} = 2 \mu s_{ij} \quad (1.3)
\]

\[
s_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (1.4)
\]

If the instantaneous velocity is expressed as \( u_i(x,t) = U_i(x) + u_i'(x,t) \) and the equations are time-averaged, the Reynolds-averaged Navier-Stokes equations (RANS) are formed.

After time-averaging, the equations take on the form

\[
\frac{\partial U_i}{\partial x_i} = 0 \quad (1.5)
\]

\[
\rho \frac{\partial U_i}{\partial t} + \rho U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( 2 \mu S_{ij} - \rho u_i' u_j' \right) \quad (1.6)
\]

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (1.7)
\]

The time-averaged equations are similar to the instantaneous equations with the instantaneous velocity replaced by the mean velocity. Also, the appearance of the term \( \rho u_i' u_j' \) in Eq. 1.6 is called the Reynolds stress tensor and is typically noted as \( \rho \tau_{ij} \). It represents the time-averaged rate of momentum transfer due to the turbulence. The term \( \rho u_i' u_j' \) is a symmetric tensor and has six independent components which indicates that six additional unknowns have been produced from the time-averaging. Adding these six unknowns to the unknown pressure and velocity components produces a total of ten
unknown quantities. Since we only have four equations this means that the system is not closed, this is generally referred to as the “closure problem”.

Additional equations can be formed by taking moments of the Navier-Stokes equations. This can be done by multiplying the Navier-Stokes equations by a fluctuating property and time averaging the product. This procedure produces a differential equation for the Reynolds stress tensor.

\[
\frac{\partial \tau_{ij}}{\partial t} + U_k \frac{\partial \tau_{ij}}{\partial x_k} = -\tau_{ik} \frac{\partial U_j}{\partial x_k} - \tau_{jk} \frac{\partial U_i}{\partial x_k} + \varepsilon_{ij} - \Pi_{ij} + \frac{\partial}{\partial x_k} \left[ \nu \frac{\partial \tau_{ij}}{\partial x_k} + C_{ijk} \right]
\]  

\(1.8\)

\[
\Pi_{ij} = \frac{\rho'}{\rho} \left( \frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i} \right) 
\]

\(1.9\)

\[
\varepsilon_{ij} = 2\nu \frac{\partial u'_i}{\partial x_k} \frac{\partial u'_j}{\partial x_k} 
\]

\(1.10\)

\[
\rho C_{ijk} = \rho u'_i u'_j u'_k + p' u'_j \delta_{ik} + p' u'_i \delta_{jk} 
\]

\(1.11\)

The turbulent kinetic energy equation can be derived by taking the trace of the Reynolds stress equation and noting that \(\Pi_{ij}\) vanishes for incompressible flow. By contracting Eq. 1.8 the kinetic energy equation becomes

\[
\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left[ \nu \frac{\partial k}{\partial x_j} - \frac{1}{2} \frac{\partial u'_i u'_j}{\partial x_j} - \frac{1}{\rho} \frac{\partial p'}{\partial x_j} \right] 
\]

\(1.12\)

\[
\varepsilon = \nu \frac{\partial u'_i}{\partial x_k} \frac{\partial u'_i}{\partial x_k} 
\]

\(1.13\)

The two terms on the left-hand side of Eq. 1.12 represent the rate of change of \(k\) following a fluid particle. The first term on the right-hand side represents the production of the turbulent kinetic energy that is due to the interaction of turbulent stress and the gradient of the mean flow velocity. The second term, \(\varepsilon\), represents the rate of dissipation of the turbulent kinetic energy occurring at the small turbulence scales. The third term represents
the molecular diffusion of the turbulent kinetic energy and the last two terms represent the
diffusion of turbulent kinetic energy from high intensity to low intensity due to turbulent
fluctuating motion. The above equation is not closed since it contains unknown quantities
for $\tau_{ij}$, dissipation, and turbulent diffusion. Closure of the $k$ equations can be achieved
by replacing the unknown correlations with closure approximations which are based on
experimental data. The Reynolds stress tensor, $\tau_{ij}$, can be replaced by assuming that the
Boussinesq approximation is valid. This assumption allows us to represent the Reynolds
stress tensor as

$$\tau_{ij} = 2\nu_t S_{ij} - \frac{2}{3} k \delta_{ij} \quad (1.14)$$

where $S_{ij}$ is the mean strain-rate tensor. The second term in Eq. 1.14 is needed to obtain
the proper trace of $\tau_{ij}$. The last two terms in Eq. 1.12 can also be thought of as turbulent
diffusion and pressure diffusion. The pressure diffusion term is usually combined with the
turbulent diffusion term and the combined term is modeled as a gradient-diffusion term

$$-\frac{1}{2} u_i' u_i' - \frac{1}{\rho} p' u_j' = \frac{\nu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \quad (1.15)$$

where $\sigma_k$ is a closure constant. Now the $k$ equation becomes

$$\frac{Dk}{Dt} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_t) \frac{\partial k}{\partial x_j} \right] \quad (1.16)$$

where the second term $\varepsilon$ still needs to be modeled. The derivation of the $\varepsilon$ equation
follows from taking the following moment of the Navier-Stokes equation:

$$2\nu \frac{\partial u_i'}{\partial x_j} \frac{\partial}{\partial x_j} [N (u_i)] = 0 \quad (1.17)$$
where \( N (u_i) \) is the Navier-Stokes operator.

\[
N (u_i) = \rho \frac{\partial u_i}{\partial t} + \rho u_k \frac{\partial u_i}{\partial x_k} + \frac{\partial p}{\partial x_i} - \mu \frac{\partial^2 u_i}{\partial x_k \partial x_k} \tag{1.18}
\]

The \( \varepsilon \) equation becomes

\[
\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_l} \left( -\varepsilon u_l - \frac{2\nu}{\rho} \frac{\partial u_i}{\partial x_j} \frac{\partial p}{\partial x_j} + \nu \frac{\partial \varepsilon}{\partial x_l} \right) - 2\nu \frac{\partial U_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_i} \frac{\partial u_l}{\partial x_l} + \frac{\partial u_i}{\partial x_l} \frac{\partial u_j}{\partial x_l} \right)
- 2\nu u_l \frac{\partial u_i}{\partial x_j} + \frac{\partial^2 U_i}{\partial x_l \partial x_l} - 2\nu \frac{\partial u_i}{\partial x_l} \frac{\partial u_i}{\partial x_l} \frac{\partial u_j}{\partial x_l} - 2 \left( \nu \frac{\partial^2 u_i}{\partial x_l \partial x_l} \right)^2
\]

where the term

\[-\varepsilon u_l - \frac{2\nu}{\rho} \frac{\partial u_i}{\partial x_j} \frac{\partial p}{\partial x_j}\]

represents the turbulent diffusion of \( \varepsilon \), the term

\[\nu \frac{\partial \varepsilon}{\partial x_l}\]

represents the molecular diffusion of \( \varepsilon \), the terms

\[2\nu \frac{\partial U_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_i} \frac{\partial u_l}{\partial x_l} + \frac{\partial u_i}{\partial x_l} \frac{\partial u_j}{\partial x_l} \right) - 2\nu u_l \frac{\partial u_i}{\partial x_j} + \frac{\partial^2 U_i}{\partial x_l \partial x_l} - 2\nu \frac{\partial u_i}{\partial x_l} \frac{\partial u_i}{\partial x_l} \frac{\partial u_j}{\partial x_l} - 2 \left( \nu \frac{\partial^2 u_i}{\partial x_l \partial x_l} \right)^2\]

represents the production of \( \varepsilon \), and the last two terms

\[2\nu \frac{\partial u_i}{\partial x_l} \frac{\partial u_i}{\partial x_l} \frac{\partial u_j}{\partial x_l} - 2 \left( \nu \frac{\partial^2 u_i}{\partial x_l \partial x_l} \right)^2\]

represent the destruction of \( \varepsilon \). The closure approximation for the turbulent diffusion term is

\[-\varepsilon u_l - \frac{2\nu}{\rho} \frac{\partial u_i}{\partial x_j} \frac{\partial p}{\partial x_j} = C_\varepsilon \left( \frac{l^2}{t} \right) \frac{\partial \varepsilon}{\partial x_j} = C_\varepsilon \frac{k^2}{\varepsilon} \frac{\partial \varepsilon}{\partial x_i}\]

where the length scale \( l \) is defined as \( l = k^{3/2}/\varepsilon \), and the time scale \( t \) is describes as \( t = k/\varepsilon \). The approximations for \( l \) and \( t \) assumes that \( k \) and \( \varepsilon \) are the correct scaling functions when in fact \( \nu \) and \( \varepsilon \) could possibly be used. The only thing that is certain about the closure approximation above is that it is dimensionally correct. The approximation is
isotropic because $C_\varepsilon k^2/\varepsilon$ is a scalar and is independent of flow direction. The production term is modeled as

$$2\nu \frac{\partial U_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_i} \frac{\partial u_i}{\partial x_j} + \frac{\partial u_i}{\partial x_i} \frac{\partial u_j}{\partial x_j} \right) - 2\nu u_i \frac{\partial u_i}{\partial x_i} \frac{\partial^2 U_i}{\partial x_i \partial x_j} = C_{\varepsilon 1} \frac{\varepsilon^2}{\varepsilon} \frac{\partial u_i}{\partial x_j} \frac{\partial U_i}{\partial x_j}$$

where $C_{\varepsilon 1}$ is a closure coefficient. The destruction term is modeled as

$$2\nu \frac{\partial u_i}{\partial x_i} \frac{\partial u_i}{\partial x_j} - 2 \left( \nu \frac{\partial^2 u_i}{\partial x_i \partial x_j} \right)^2 = -C_{\varepsilon 2} \frac{\varepsilon^2}{k}$$

The $\varepsilon$ equation contains considerably more double and triple correlations than the $k$ equation and these correlations are essentially impossible to measure using experiments. This means that the use of DNS is the only way to obtain accurate predictions of these correlations. Chen [2] show the comparison of the above closure approximations with DNS data of Moser [3] for flow in a channel at Reynolds number $Re_\tau = 180$. The results show that the closure approximations only marginally match the DNS results. Chen [2] also shows that when the summation of modeled production and destruction with respect to the exact production, triple correlation, and destruction terms, the curve matches the DNS results fairly well. Chen [2] indicates that the summation of errors is an order of magnitude smaller than the error of each term. This might be the case for flow in a channel but it is certainly unlikely that this behavior would be observed for more complex flows. The final form of the $\varepsilon$ equation is

$$\frac{D\varepsilon}{Dt} = C_{\varepsilon 1} \frac{\rho U_i}{k} \frac{\partial U_i}{\partial x_j} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right]$$

where $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, and $\sigma_\varepsilon$ are closure coefficients.

1.2 Outline of the Thesis

Chapter 1 introduces the approximations used to develop Reynolds averaged two-
equation turbulence models. In Chapter 3, several of the well known two-equation models have been applied to predict the flow inside of a stirred tank reactor. All of the models underpredict the spreading rate of the impeller discharge region causing the circulation rate above and below the impeller to be overpredicted. All of the models underpredict the kinetic energy near the shaft which is due to their inability to accurately represent the destabilization of turbulence associated with streamline curvature. Conventional two-equation models require damping functions to accurately predict near wall turbulence. These damping functions are shown to be ill suited for complex flows, such as the flow in a stirred tank, causing the models to produce poor near wall turbulence predictions. In Chapter 4 an elliptic relaxation approach is introduced which is used to develop a new model (the $v^2f - k\omega$ model) that does not require near wall damping. In Chapter 5 the details of the flow solver are introduced. In Chapter 6 this new model is compared with other two-equation models including Durbin’s [1] $v^2f - k\varepsilon$ model, for predicting several different flows which have been well studied and have detailed experimental data. In Chapter 7, the models are applied to predict a jet-in-a-crossflow. In Chapter 8 concluding remarks are given along with the goals for future work.
Chapter 2. Conventional Turbulence Models

2.1 Two-Equation Models

Two-equation models have served as the foundation for computing turbulent flows during the last two decades. For all two-equation models, the starting point is the Boussinesq approximation, Eq. 1.14, and the turbulence kinetic energy equation, Eq. 1.12. The standard approximation for turbulent transport of a scalar quantity is that of gradient-diffusion. In analogy to molecular diffusion we say that

\[ -u_j \phi = \nu_t \partial \Phi / \partial x_j. \]

Since there is no corresponding analogy for the pressure-diffusion term, it is typically grouped with the turbulent transport term and the sum is assumed to behave as a gradient diffusion process. With this approximation we end up with Eq. 1.16.

The term \( \varepsilon \) is a modeled term which has been the primary focus of researchers and several definitions have been developed. Prandtl [4] developed the following expression for the turbulence dissipation, \( \varepsilon \), where \( l \) is the turbulent length scale.

\[ \varepsilon = C_D k^{3/2} / l \]  

(2.1)

To describe the length scale, \( l \), Prandtl [4] used the mixing length model which means that \( l \propto l_{mix} \). This approximation only holds if the ratio of production to dissipation is constant. With such a restrictive assumption, the Prandtl mixing length model can only be used for the simplest of flows. Kolmogorov [5] pointed out that a second transport equation is needed to compute \( \omega \), which is the specific dissipation rate having units of \((\text{time})^{-1}\). Chou [6] proposed modeling the exact equation for \( \varepsilon \). Based on these ideas several different versions of the \( k-\varepsilon \) and \( k-\omega \) two-equation models have been developed.
A major disadvantage of using these models is that they can not accurately predict near wall turbulence. To improve near wall predictions, additional near wall modification must be made to the models. There are two basic types of near wall modifications that have been used to date. The first of these modifications is the use of wall functions. In this approach the turbulence equations are not solved down to the wall, but instead a matching procedure is used to prescribe the velocity and turbulence quantities at the first grid point from the wall. This approach uses the law of the wall as the constitutive relation between velocity and the wall shear stress. The law of the wall is formulated as

\[ U = u_r \left[ \frac{1}{\kappa} \ln \left( \frac{u_{\tau}y}{\nu} \right) + C \right] \]  

(2.2)

where \( \kappa = 0.41 \) and \( C = 5.0 \). Dividing Eq. 2.2 by \( u_r \) we get the following expression:

\[ u^+ = \frac{1}{\kappa} \ln \left( \frac{u_{\tau}y}{\nu} \right) + C \]

which is used to prescribe the velocity at the first grid point off the wall depending upon the value of \( y^+ \) at that point. The turbulence quantities must also be prescribed at the first point off the wall using the relations:

\[ k = \frac{u_r^2}{\sqrt{\beta_o}}, \quad \omega = \frac{k^{1/2}}{(\beta_o^{1/4})^{1/4}}, \quad \varepsilon = \frac{(\beta_o)^{3/4} k^{3/4}}{\kappa y} \]

The wall function approach is not always applicable for several reasons. One problem with using this approach is that the solution is now dependent upon the location of the first grid point from the wall. Also, this approach is only applicable for wall bounded flows where separation does not occur. Another alternative to using wall functions is to use low Reynolds number modeling in which "damping functions" are incorporated into the transport equations so that the near wall turbulence is modelled accurately. The use of
low Reynolds number modeling eliminates the need to make assumptions about the nature of the turbulence or velocity profile near solid walls. There are many different forms of "damping functions" which have been developed by researchers over the years. One thing that they all have in common is that they are formulated in terms of one or both of the following parameters.

\[ \text{Re}_t = \frac{k^2}{\varepsilon \nu}, \quad \text{Re}_y = \frac{k^{1/2} y}{\nu} \]

An advantage to using low Reynolds number modeling is that the transport equations for the turbulence quantities can be solved down to the wall providing better predictions for near wall turbulence in complex flows. The disadvantage is that the "damping functions" used in the low Reynolds number modeling are typically ill suited for complex flows. Despite this, low Reynolds number modeling is still a better alternative to the wall function approach when computing complex flows. At this point some of the different versions of the \( k - \varepsilon \) and \( k - \omega \) model will be discussed. Although there are several different versions of these models, only the versions of the \( k - \varepsilon \) and \( k - \omega \) models which were used in this study to predict the flow in stirred tank reactors, is described in this chapter.

2.1.1 \( k - \varepsilon \) Model (LKE)

The \( k - \varepsilon \) model is the most popular of the two-equation models and has produced qualitatively satisfactory results for a number of complex flows. The low Reynolds number \( k - \varepsilon \) (LKE) model solves a transport equation for turbulent kinetic energy \( k \) and a transport equation for the isotropic part of dissipation \( \tilde{\varepsilon} \) which, unlike \( \varepsilon \), goes to zero at the wall. Damping functions \( f_\mu, f_1, f_2, \) and \( f_3 \) of Rodi [7] are used to account for the
near-wall effects. Note that Rodi [7] used Direct Numerical Simulation (DNS) data to curve-fit the model coefficients and expressions, and have presented their model as an improvement over other competing models. The incompressible form of the equations are:

\[
\nu_t = C_\mu f_\mu \frac{k^2}{\bar{\varepsilon}} \tag{2.3}
\]

\[
\frac{Dk}{Dt} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \bar{\varepsilon} + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_t) \frac{\partial k}{\partial x_j} \right] \tag{2.4}
\]

\[
\frac{D\bar{\varepsilon}}{Dt} = C_{\varepsilon_1} f_1 \frac{\bar{\varepsilon}}{k} \tau_{ij} \frac{\partial u_i}{\partial x_j} - C_{\varepsilon_2} f_2 f_3 \frac{\bar{\varepsilon}^2}{k} + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_t) \frac{\partial \bar{\varepsilon}}{\partial x_j} \right] \tag{2.5}
\]

with \( \bar{\varepsilon} = \varepsilon - D, \delta_{ij} \) is the Kronecker delta functions and

\[
\tau_{ij} = \nu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \tag{2.6}
\]

The near-wall damping functions and model constants are defined by:

\[
f_\mu = 1 - \exp \left( -0.0002 \text{Re}_y - 0.00065 \text{Re}_y^2 \right), \quad f_1 = 1.0, \quad \sigma_\varepsilon = 1.0,
\]

\[
f_2 = 1 - 0.3 \exp \left( - \text{Re}_t^2 \right), \quad \sigma_k = 1.3, \quad C_{\varepsilon_1} = 1.44, \quad f_3 = \exp \left( 2R_p^3 \right),
\]

\[
C_{\varepsilon_2} = 1.92, \quad C_\mu = 0.09, \quad R_p = \frac{P/\varepsilon}{0.3 \sqrt{\text{Re}_t}}, \quad \text{Re}_t = \frac{k^2}{\nu \varepsilon}, \quad \text{Re}_y = \frac{k^{1/2}y}{\nu}
\]

where \( P \) is the production of turbulence and \( \text{Re}_t \) and \( \text{Re}_y \) are turbulence Reynolds numbers. The functions \( E \) and \( D \) represent near-wall effects and are defined by:

\[
E = 2\nu \nu_t \left( \frac{\partial u_i}{\partial x_j} \right)^2, \quad D = 2\nu \left( \frac{\partial k^{1/2}}{\partial x_j} \right)^2
\]

Note that in this model the introduction of the \( f_3 \) and the curve-fitted expression for \( f_\mu \) are based on DNS data, and according to Rodi [7] represent improvements over the conventional flow \( \text{Re} k - \varepsilon \) models. The constant 0.3 appearing in the expression was based on a sensitivity study and appeared to be the best choice.
2.1.2 Modified $k - \varepsilon$ Model (MKE)

The modified $k - \varepsilon$ (MKE) model is similar to the above LKE model except that the $\varepsilon$ equation has been modified using the gradient Richardson number correction. This modification is imposed in order to improve swirling flow predictions. The new $\varepsilon$ equation becomes:

$$\frac{D\varepsilon}{Dt} = C_{\varepsilon 1} f_1 \frac{\varepsilon}{k} \tau_{ij} \frac{\partial u_i}{\partial x_j} - C_{\varepsilon 2} f_2 f_3 \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \sigma \nu_t \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial \varepsilon}{\partial x_j} \right]$$ \hspace{1cm} (2.7)

where

$$C_{gs} = 0.02, \quad Ri_{gs} = \frac{k^2 u_{\theta} \partial (ru_{\theta})}{\varepsilon^2 r^2 \partial r}$$

and $u_{\theta}$ is the tangential component of velocity.

The gradient Richardson number represents the ratio of an apparent body force acting on a fluctuating, or displaced fluid element, to a typical inertial force Sloan [8]. A positive gradient will tend to produce a positive Richardson number which will increase the dissipation rate and decrease the kinetic energy and eddy viscosity. A negative gradient will tend to produce a negative Richardson number and increase the kinetic energy and eddy viscosity.

2.1.3 RNG $k - \varepsilon$ Model (RNG)

The RNG-based $k - \varepsilon$ (RNG) model follows the same framework as the other two equations models but uses Renormalization Group methods, Yakhot [9]. The model is said to provide improved predictions of near-wall flows and flows with high streamline curvature. It has also shown to improve results for highly swirling flows. The governing
equations and the two-layer wall function model are:

\[
\mu_{eff} = \mu_{mol} \left[ 1 + \sqrt{\frac{C_\mu}{\mu_{mol}} \frac{k}{\sqrt{\varepsilon}}} \right]^2
\]  
(2.8)

\[
\frac{Dk}{Dt} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left[ \sigma_k \mu_{eff} \frac{\partial k}{\partial x_j} \right]
\]  
(2.9)

\[
\frac{D\varepsilon}{Dt} = C_{\varepsilon 1} \frac{\varepsilon}{k} \tau_{ij} \frac{\partial u_i}{\partial x_j} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \sigma_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right] - R
\]  
(2.10)

where \( R \) in the \( \varepsilon \) equation is given by:

\[
R = \frac{C_\mu \eta^3 \left( 1 - \frac{\eta}{\eta_o} \right) \varepsilon^2}{1 + \beta \eta^3 k}
\]  
(2.11)

with \( \eta = Sk/\varepsilon \) and \( S \) is the modulus of the mean rate-of-strain tensor, \( \eta_o = 4.38 \), \( \beta = 0.012 \). The model constants are:

\[
C_{\varepsilon 1} = 1.42, \quad C_{\varepsilon 2} = 1.68, \quad \sigma_k = 1.393, \quad \sigma_\varepsilon = 1.393
\]

The two-layer zonal model splits the domain into a viscosity-affected region and a fully turbulent region. The two regions are divided by the turbulent Reynolds number, \( Re_y \).

In the viscosity-affected region (\( Re_y < 200 \)), the one equation model of Wolfstein [10] is used. In the one-equation model the momentum and \( k \) equations are solved but \( \varepsilon \) and the eddy viscosity \( \nu_t \) are computed using the following expressions.

\[
\varepsilon = \frac{k^{3/2}}{l_\varepsilon}, \quad \nu_t = C_\mu \sqrt{k} l_\mu
\]

The length scales \( l_\varepsilon \) and \( l_\mu \) are defined as:

\[
l_\varepsilon = c_l y \left[ 1 - \exp \left( -\frac{Re_y}{A_\varepsilon} \right) \right], \quad l_\mu = c_l y \left[ 1 - \exp \left( -\frac{Re_y}{A_\mu} \right) \right]
\]

\[
c_l = \kappa C_\mu^{-3/4}, \quad A_\varepsilon = 70, \quad A_\mu = 2 c_l
\]

2.1.4 \( k - \omega \) Model (LKW),(HKW)

For both the LKW and HKW models \( k = 0 \) on solid boundaries. For \( \omega \), the rough wall
boundary condition of Wilcox [11] was used at the wall, \( \omega_w = 2500 \times \nu_w / \kappa_s^2 \). Here \( \kappa_s \) is a roughness factor to be computed. The resulting equations for \( k, \omega, \) and \( \nu_t \) are:

\[
\begin{align*}
\nu_t &= \alpha^* \frac{k}{\omega} \\
\frac{Dk}{Dt} &= \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \omega k + \frac{\partial}{\partial x_j} \left[ \left( \nu + \sigma_k \nu_t \right) \frac{\partial k}{\partial x_j} \right] \\
\frac{D\omega}{Dt} &= \frac{\sigma}{\nu_t} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \nu + \sigma_\omega \nu_t \right) \frac{\partial \omega}{\partial x_j} \right] + 2\sigma_\omega^2 \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}
\end{align*}
\]

(2.12)

The model constants for the HKW model are:

\[
\begin{align*}
\sigma_k &= 0.5, \quad \sigma_\omega = 0.5, \quad \beta = 0.075, \quad \beta^* = 0.09, \quad \kappa = 0.41, \quad \alpha = 5/9, \\
\alpha^* &= 1.0, \quad \sigma_\omega^2 = 0.75
\end{align*}
\]

For the LKW model the constants are:

\[
\begin{align*}
\beta^* &= \frac{9}{100} \frac{5}{18} + \left( \frac{Re_t}{R_{\beta}} \right)^4, \quad \alpha^* = \frac{\alpha_o^*}{1 + \left( \frac{Re_t}{R_{\beta}} \right)^4}, \quad \alpha = \frac{5}{9} \frac{\alpha_o + \frac{Re_t}{R_k}}{1 + \left( \frac{Re_t}{R_k} \right)^4} (\alpha^*)^{-1} \\
\beta &= 0.075, \quad \alpha_o = 0.1, \quad \sigma_k = 0.5, \quad \sigma_\omega = 0.5, \quad R_{\beta} = 8 \\
R_k &= 6, \quad \alpha_o^* = \beta/3, \quad R_\omega = 2.7
\end{align*}
\]

The HKW and LKW models both integrate the transport equations for \( k \) and \( \omega \) all the way to the wall, but the HKW model has been shown to be asymptotically inconsistent with the expected behavior of \( k \) and dissipation, \( \varepsilon = \beta^* \omega k \), approaching a solid boundary. Also, the HKW model, like most two-equation models, predicts transition from laminar to turbulent at a critical Reynolds number, \( Re_c \approx 8100 \), which is much lower than the minimum critical Reynolds number, \( Re_c \approx 90,000 \), which Wilcox derived using linear-stability theory of a Blasius boundary layer. The LKW model tries to remedy these problems by using functional closure coefficients instead of constant coefficients.
2.1.5 Modified $k - \omega$ Model (MKW)

The modified $k - \omega$ (MKW) model is similar to the above LKM model except that the $\omega$ equation has been modified using the gradient Richardson number and a correction formulated by Bardina (cited in Reynolds [12].) Both of these corrections are designed to enhance dissipation in the presence of stabilizing curvature. A more detailed discussion of these corrections will be given in the next section. The new $\omega$ equation becomes:

$$
\frac{D\omega}{Dt} = \frac{\sigma \nu_t}{\nu_t} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta (1 - C_{gs} Ri_{gs}) \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \nu + \sigma \omega \nu_t \frac{\partial \omega}{\partial x_j} \right) \right] + 2 \sigma \omega^2 \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} - C_B k \omega \zeta
$$

(2.13)

where

$$C_B = 3, \quad Ri_{gs} =, \quad C_{gs} =,$$

$$\zeta = \left[ \left( \frac{\partial u_\theta}{\partial z} \right)^2 + \left( \frac{\partial u_\theta}{\partial r} + \frac{u_\theta}{r} \right)^2 + \left( \frac{\partial u_r}{\partial z} - \frac{\partial u_a}{\partial r} \right)^2 \right]$$

(2.14)

In the above expression $u_r$ and $u_a$ are the radial and axial components of velocity.

The Bardina correction is based on the assumption that rotation appears to trap the energy in the large scales, organize the fluctuations into more coherent eddies, and decrease the cascade of energy transfer to the small eddies. The correction is defined as $\zeta$ and is the square root of the scalar product of the mean vorticity vectors contracted with itself.
Chapter 3. Stirred Tank

Computation of turbulent flow in an impeller stirred tank reactor (STR) can be a considerable challenge for existing turbulence models. Factors contributing to this difficulty include the non-isotropic nature of the flow in a stirred tank, the complex geometry of rotating impellers and the large disparity in geometric scales present. In addition, the flow and turbulence encountered and produced by each blade are further complicated due to the fact that the blade itself is riding in the wake of another blade.

Past work with modeling turbulent flow in STRs has been focused primarily on simplified computational analyses. This is not surprising since the problem is quite complex geometrically and the flow is unsteady. By far, the most popular simplified analysis has been to use experimentally measured velocity and kinetic energy profiles at the impeller tip as boundary conditions to approximate the impeller. In this approach, the tank wall is the only solid surface modeled. This simplified approach neglects the impeller geometry and models the impeller region as a fictitious disk where transport variables are input as inflow/outflow boundary conditions. A multitude of applications employing this technique have been reported in the literature. A few include Harvey [13], Ranade [14], Ju [15], Kresta [16], Bakker [17], and Ducoste [18]. Many other numerical results obtained using the technique have been reported throughout the literature in the past few years.

Past attempts to eliminate the experimental (or empirical) input in the CFD calculation have been primarily through the solution of the high Reynolds number (Re) $k - \varepsilon$ transport equations and the impeller and tank surfaces. Use of this technique eliminates the
numerical difficulties associated with the low-Re $k - \varepsilon$ models where the $k$ and $\varepsilon$ equations are integrated all the way through the viscous sublayer to the surface of a solid wall.

Typical industrial sized STRs are large and often contain multiple impeller configurations. The grid resolution required for the application of low-Re turbulence models can be excessive and therefore high-Re turbulence models with wall functions are typically used in simulations performed by industry.

While wall functions in the high-Re models alleviate the problems with grid resolution, their applicability may be questioned under certain flow conditions. As an example, at impeller surfaces, boundary layers are not fully developed before trailing edge separation occurs, and the use of wall functions in these regions of the tank is clearly questionable. Dong [19], presents results using a popular commercial CFD package (FLUENT) and the standard $k - \varepsilon$ model with wall functions to model the flow resulting from a paddle impeller in an unbaffled tank. The model predicts the overall features of the flow satisfactorily. However, the tangential velocities are underpredicted near the shaft and the impeller discharge velocities are significantly overpredicted.

Harvey [20] have focused exclusively on modeling laminar flow in STRs. Their approach is to represent the geometry precisely using a generalized coordinate system, eliminating the need for experimental input of boundary conditions. Wechsler [21], have extended this technique to turbulent flows and it is used in the present work. Six two-equation turbulence models are applied to the flow generated by a paddle impeller in an unbaffled tank and investigated experimentally by Dong [22]. The experimental
configuration consists of a single set of eight blades with a rotation speed of 100 rpm. This corresponds to a Reynolds number (based on impeller diameter and the mean velocity at the tip of the impeller) of 3273. The goal of this study is to provide a detailed investigation of the predictive capabilities of two-equation turbulence models to predict the flow in STRs. The flow field and turbulence quantities have been computed throughout the tank and ad hoc models for the impeller region have not been used. As a first step, an un baffled tank is considered to avoid the complexities induced by side wall baffles.

Note that only the Reynolds-averaged Navier-Stokes equations are solved to provide predictions of the time-averaged velocity and turbulence quantities. The effect of velocity fluctuations due to turbulence or large scale unsteady structures on the time-averaged quantities, is essentially represented by the turbulence model for the Reynolds stresses. The accuracy of the predictions of the mean statistics depends on how well these models capture the effect for the flow fluctuations (over the entire spectrum of scales).

The turbulence models considered in this study include the low Reynolds number (LKW) and high Reynolds number (HKW) \( k - \omega \) models of Wilcox [11], a low Reynolds number \( k - \varepsilon \) model (LKE), RNG \( k - \varepsilon \) model (RNG), and modified \( k - \omega \) (MKW) and \( k - \varepsilon \) (MKE) models which use a correction for streamline curvature and swirl.

The LKW, HKW, and the MKW models consist of transport equations for turbulent kinetic energy, \( k \), and specific turbulent dissipation, \( \omega \). The low Reynolds number \( k - \varepsilon \) model (LKE), modified \( k - \varepsilon \) model (MKE), and the RNG \( k - \varepsilon \) (RNG) consist of transport equations for turbulent kinetic energy, \( k \), and turbulent dissipation \( \varepsilon \). The values
of \( k \) and \( \omega \) or \( \varepsilon \) are then used to compute the eddy viscosity, \( \nu_t \). Low Reynolds number models require ad hoc damping functions at a solid boundary to insure the correct profile for the eddy viscosity near the boundary. The LKE, LKW, and MKE models integrate the transport equations for turbulence quantities all the way to the impeller surfaces. Thus, the boundary layer and its spatial growth rate are computed as part of the solution. The value of \( y^+ \) at the first grid point off the wall was less than 1.0 for all of the computations. The RNG \( k - \varepsilon \) model uses a two-layer zonal model for computing \( \varepsilon \) near solid boundaries.

It should be noted that the present study is the first comparative study of the application of low-Reynolds number turbulence models (LKE, LKW, MKE, MKW) to STRs. Further, in the MKE and MKW models we have investigated the effect of adding specific swirl and curvature corrections. The application to STRs in the low-Reynolds number formulation are also unique.

In the next section, a brief overview of the time-averaged Navier-Stokes equation solver is presented followed by an overview of each of the turbulence models. In the results section, the models are tested for the flow in the stirred tank studied experimentally by Dong [22].

3.1 Mean Flow Equations

The mean flow equations are the steady incompressible Navier-Stokes equations written in a generalized coordinate system. These equations are solved in a rotating frame of reference, and therefore, centrifugal and Coriolis force terms appear in the momentum equations.
The coupling between the velocity and pressure fields is accommodated through the use of the pseudocompressibility technique Rogers [23] which introduces a pseudo-time derivative of pressure in the continuity equation. Solutions to the resulting hyperbolic equation set are obtained by marching in pseudo-time and driving the pressure derivative to zero. Details on the flow equations and their numerical solutions can be found in Rogers [23]. Third-order upwind differencing was used for the convective terms and second-order central differencing was used for the viscous terms.

3.2 Results and Discussion

3.2.1 Reactor Geometry and Operating Conditions

The flow investigated experimentally by Dong [22] is chosen in the present work to study the performance of the turbulence models for flow in stirred tank reactors. These experiments are chosen primarily due to the absence of baffles in the experiments, which greatly simplifies the computational problem. Thus, the turbulence models can be evaluated without the need for simulations involving sliding meshes. Detailed measurements of all three components of the mean velocity and the rms velocity is reported in the paper by Dong [22] for two different geometry configurations. For both configurations the tank is unbaffled with a diameter/(tank height) ratio of 1. Both tank configurations have been simulated in the present work and will be referred to as case 1 and case 2. In case 1, a 2.5 cm. diameter paddle impeller is positioned at mid-height in the tank. In case 2, the impeller is positioned at 1/3 of the tank height from the bottom of the tank. These configurations, along with the remaining dimensions, are shown
schematically in Fig. 3.1. In both cases the impeller rotational speed is 100 rpm and the kinematic viscosity, $\nu = 10^{-6} m^2/s$. This corresponds to a Reynolds number, $Re = 3273$. The computational results were circumferentially averaged and compared with the time averaged data of Dong [22]. In the experiments, measurements of mean and fluctuating velocity were taken using a sampling rate which was much greater than the frequency of a blade passage time. Circumferential averaging is equivalent to time averaging in STR flows without baffles since each circumferential location represents the blade location at some instant in time. The calculations were performed on a three-dimensional computational domain using a fixed grid consisting of an overlapping multi-block grid topology. The grids used for case 1 and case 2 are shown in Fig. 3.2. The computational domain consists of one of the eight regions between neighboring blades of the impeller. This $\pi/4$ circumferential slice is discretized into six different grid zones, each of which overlaps with its neighboring zones using a Chimera approach. Zone 1 describes the
Figure 3.2. Topology and computational grid for case 1 and case 2

region directly between the two impeller blades. Zones 2 and 3 sit directly underneath and overhead, respectively, of the impeller zone (zone 1). Zone 5 represents the discharge region of the impeller and is made finer than the surrounding zones. Zones 4 and 6 extend from the impeller blade tips out to the tank wall and sit underneath and overhead, respectively, of zone 5. Boundary conditions for faces of zones, which are not physical boundaries of the domain, are obtained using a trilinear interpolation procedure from points interior to neighboring zones. The trilinear interpolation is applied at the Chimera interfaces to update the solution.

A primary reason for the selection of this particular grid topology is that clustering of the grid at all solid surfaces can take place efficiently without the need for propagating the clustering into the far-field. In addition, specific regions of the grid can be made finer than other regions. This topology can be stacked end-on-end for multiple impeller applications.
A grid independence study was performed using the MKE model for case 1. The tangential and radial velocity profiles are presented in Fig. 3.3 for coarse, medium, and fine grid solutions and indicate that the computed results are grid independent. Note that in going from the coarsest grid to the finest grid, the number of grid points has been increased nearly four times (94,500 – 338,000 points). Different levels of grid refinement have been implemented in different zones, with the highest refinement in the regions with the largest gradients. The results in this study are based on the fine grid computations.

3.2.2 Numerical Results

Kresta [24] has given a clear overview of the turbulence characteristics and some modeling challenges associated with STRs. In the papers by Kresta [25], Fort [26], Zhou [27], and Jaworski [28] it is shown that all three time averaged rms velocity components are approximately equal close to the blades, and the assumption of local isotropy is valid. Experiments by Hockey [29] show that in the outer impeller region away from the blade this is not the case. Experiments by Brodkey [30] indicate that local isotropy may exist if the local Reynolds number \( Re_\lambda \) (based on the turbulent length scale \( \lambda \) and the fluctuating velocity) is greater than 800, while Sreenivasan [31] indicates that local isotropy may exist for \( Re_\lambda = 50 \). Kresta [24] also indicates that in the impeller region \( Re_\lambda \) is approximately 200–400 for typical experimental conditions. In the remainder of the tank, \( Re_\lambda \) is approximately 60–150. Thus, regions of the flow near the impeller blades may exhibit isotropy, while other regions may be anisotropic, and the present results will shed light on the ability of the two equation models to accurately predict such a flow field.
Figure 3.3. Velocity profiles for grid independence study
It is also uncertain if conventional two-equation models can capture the dynamics associated with the vortex shedding from the blades. Stoots [32] have shown that for a Rushton blade these vortices are well defined and have a diameter of order $D/10$, where $D$ represents the diameter of the impeller. The current blade configuration is a paddle blade which produces similar flow characteristics as the Rushton impeller. It is believed these vortices have a large influence on the turbulence energy and dissipation in the impeller region. Since traditional turbulence models are calibrated against simpler flows, their performance in more complex flows have been mixed.

Large scale, low-frequency motions in the flow have also been investigated by Kresta [33]. The experiments show that the pitch-blade turbines are more susceptible to these low-frequency motions than are Rushton blade turbines. As stated before, the present tank configuration is a paddle blade which exhibits flow characteristics similar to the Rushton impeller, and therefore these low-frequency motions are unlikely to be important in the present flow.

Figure 3.4 shows the computed mean velocity field for both cases using the modified $k - \varepsilon$ (MKE) model. Predictions for all six models studied look qualitatively similar. The predicted flow field is dominated by a radial jet emanating from the impeller which produces two major circulating regions in the tank, one above and one directly below the impeller. The two recirculating eddies are roughly symmetrical in Fig. 3.4, but the presence of the lower boundary surface leads to considerable asymmetry in case 2. The
bottom wall causes the radial jet in case 2 to be slightly pointing downward increasing the circulation rate of the lower recirculating eddy. This also causes the lower eddy to be placed closer toward the shaft compared to the upper eddy. In Fig. 3.5 the tangential velocity component is compared with the experiment for cases 1 and 2. The tangential velocity in the impeller discharge region is accurately predicted by each model. However, near the shaft, above and below the impeller, the tangential velocity is severely underpredicted. It is also apparent that the $k - \omega$ model predictions near the shaft are better than the $k - \varepsilon$ model predictions. This is presumably associated with the additional overprediction of the discharge velocity in the impeller region in the $k - \omega$ model predictions. In STR flows the angular momentum decreases in the radial direction and the flow has features similar to that of a free vortex (where swirl enhances turbulence).
Figure 3.5. Computed tangential velocity profiles, case 1 (left), case 2 (right)
The recirculating flow leads to streamlines with concave curvature in the \(\xi - \eta\) plane near the shaft, which destabilizes turbulence. These complex effects are not properly represented by the turbulence models, which partly contributes to the poor performance observed.

Damping functions, which are needed by most two-equation models to accurately predict near wall turbulence, are formulated for wall bounded flows or flows with moderate separation. For wall bounded flows, the turbulent Reynolds number, \(\text{Re}_t = k^2/\nu\varepsilon\), is significant in the boundary layer region close to the wall and negligible in the freestream region away from the walls. In more complex flows such as the flow in stirred tanks, \(\text{Re}_t\) can vary in magnitude along the tank walls and in the outer flow field. In these more complex flows the traditional damping functions are ill-suited and near wall turbulence may not be computed accurately. Near the shaft, the computed \(\text{Re}_t\) is not very large; thus, low Reynolds number effects in this region may not be represented properly. It seems as though all of the models, except for the HKW model, predict relaminarization of the flow near the shaft, which is not shown by experiments. This would cause the tangential velocity near the shaft to be underpredicted as observed in the current calculations. In the outer tank regions, above and below the impeller discharge, the tangential velocity is overpredicted. This is caused by an overprediction of the eddy viscosity in the outer region of the tank. The streamline curvature near the outer tank wall stabilizes turbulence. Since the models cannot represent these complex effects the eddy viscosity is overpredicted.

Figure 3.6 shows the computed radial velocity profiles compared with experimental
Figure 3.6. Computed radial velocity profiles, case 1 (left), case 2 (right)
data for cases 1 and 2. We can see that at $z/r_b = 4$ (impeller discharge region for case 1) and at $z/r_b = 2.4$ (impeller discharge region for case 2), the radial component of velocity has been overpredicted by each model. This overprediction of radial velocity is caused by an underprediction of the eddy viscosity in this region, which leads to a reduction of the lateral spreading rate of the impeller discharge. The experiments of Dong [22] show anisotropic turbulence near the shaft and in the impeller discharge region for case 1 and case 2. The curvature induced modifications to the $\varepsilon$ equation and the correction to the $\omega$ equation appear to move the radial velocities in the right direction. The MKE model seems to show more of an overall improvement to the predictions for case 2 than for case 1, especially in the impeller discharge region. This is probably a result of the fact that the experiments of Dong [22], show that the level of anisotropy in this region for case 2 is slightly less than in case 1. At other axial locations, the radial velocities are considerably smaller in magnitude, and the agreement with the predictions is more satisfactory.

The predictions and experimental data for the axial velocity are shown in Fig. 3.7. The experimental data indicates that the impeller discharge is projected slightly downwards. The models do not seem to predict this downward projection accurately. Dong [19] indicated that the flow condition in the impeller stream and close to the rotor shaft changes rapidly downstream so that relaxational effects and stress convection may be important in the downstream region. Two equation models cannot correctly account for these effects. This is reflected by the significant overprediction in the axial velocity components. The extent of overprediction in Fig. 3.7 is directly correlated with the overpredictions of the
Figure 3.7. Computed axial velocity profiles, case 1 (left), case 2 (right)
Figure 3.8. Kinetic energy profiles, case 1 (left), case 2 (right)
radial velocities seen in Fig. 3.6. The LKW model shows the highest level of axial velocity overprediction. It should be noted that the $k - \varepsilon$ models show lower levels of overprediction.

The turbulence kinetic energy profiles are shown in Fig. 3.8 for case 1 and case 2. As noted earlier, the predictions in the impeller jet region and near the shaft are rather unsatisfactory, and reflect the inability of the damping functions to properly represent low Reynolds number effects near solid walls. Inaccurate turbulent predictions by two-equation models can also be caused by improper representation of the streamline curvature. Two-equation models have been known to fail to reproduce, even qualitatively, several important features of swirling flows such as velocity component decay, jet spreading rate or diffusion rate, degree of entrainment, and kinetic energy levels. Near the impeller shaft in a stirred tank, the flow is highly curved and if the models are not suitably corrected for curvature then they will not produce accurate results.

In the present study, the gradient Richardson number is used in the modified $k - \varepsilon$ model to account for swirl effects, while both the gradient Richardson number and $\zeta$-correction are used together in the modified $k - \omega$ model. It was found that the $\zeta$ correction helped to increase the kinetic energy in the impeller region of the tank and the gradient Richardson number correction increased the kinetic energy near the shaft. From Fig. 3.8 we see that the LKE model predicts the kinetic energy fairly accurately in the impeller discharge region. The kinetic energy near the shaft is not accurately predicted which motivates the present use of the gradient Richardson number correction in Fig. 2.14.
leading to the MKE model. Some improvements in the shaft region are noted in the MKE predictions, but the $k$-values in the impeller discharge region are now overpredicted.

The LKW and HKW models underpredict the kinetic energy in the impeller discharge region and near the shaft. The MKW model therefore incorporates the gradient Richardson number and $\zeta$-correction together. Although these corrections improved the kinetic energy predictions slightly, they did not show a significant global improvement to the results. These corrections are not universal and have been formulated for simple turbulent flows, not for complex flows such as the flow in stirred tanks.

Ducoste [18] report that vortices trailing from the impeller blades affect the local energy dissipation in the impeller region. In their simulations, they contribute this to their poor predictions of dissipation. We believe that the trailing vortices also contribute to the poor predictions of kinetic energy in the impeller region for both case 1 and case 2. Ducoste [18] do not observe this because they have modeled the impeller region as a disk with experimental velocity and kinetic energy profiles as boundary conditions in this region. Thus their experimental profile for kinetic energy has already accounted for the additional dynamics of the trailing vortices, but they do not have experimental profiles for dissipation. In our simulations there are no ad hoc modeling of the impeller region, rather the velocity and turbulent quantities at the impeller discharge are computed in the same manner as in the bulk of the tank.

Figure 3.9 shows the eddy viscosity computed using the LKE, the MKE, the LKW, and the MKW models. The LKE model shows high values above and below the impeller
Figure 3.9. Eddy viscosity contours, (a) LKW, (b) MKW, (c) LKE, (d) MKE
blades. Relatively lower values of the eddy viscosity are observed adjacent to the shaft and in the impeller region. The low values adjacent to the shaft are responsible for the underprediction of the tangential velocities by the LKE model. The curvature corrections, represented by the MKE model, appear to appropriately enhance the eddy viscosity in the impeller discharge region as well as the region around the shaft. However, the eddy viscosity in other regions appears to be enhanced as well. The LKW model also shows similar eddy viscosity trends as the LKE model, except that the eddy viscosity values are somewhat smaller than those predicted by the LKE model. The MKW model, incorporating both Richardson number and $\zeta$ corrections, significantly enhances the eddy viscosity in the impeller discharge region. This explains the significant improvements in the radial velocities observed with the MKW model.

It is difficult to determine which model produces better predictions. If the impeller region is not predicted accurately, then the models will produce poor predictions in the remainder of the tank. For this reason, most researchers have prescribed experimental boundary conditions for the impeller region. Sahu [34] have shown that if experimental boundary conditions are prescribed, then two-equation models will accurately predict the flow in the bulk of the tank. Therefore, one way to quantify the error between models is by analyzing the difference between the model predictions and experiments in the impeller region. Tables 3.1 and 3.2 contain the rms error of the results in the impeller region for both case 1 and case 2, respectively. The turbulent kinetic energy and mean velocity are compared with the mean kinetic energy in the impeller region.
Table 3.1 Residual error (case 1, $z/r_b = 4$)

<table>
<thead>
<tr>
<th>Model</th>
<th>$u_r$</th>
<th>$u_z$</th>
<th>$u_{\theta}$</th>
<th>$k$</th>
<th>$k_{mean}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>HKW</td>
<td>0.1638</td>
<td>0.0674</td>
<td>0.0612</td>
<td>0.0272</td>
<td>0.0270</td>
</tr>
<tr>
<td>LKW</td>
<td>0.3267</td>
<td>0.0385</td>
<td>0.0502</td>
<td>0.0266</td>
<td>0.0280</td>
</tr>
<tr>
<td>LKE</td>
<td>0.2873</td>
<td>0.0343</td>
<td>0.0382</td>
<td>0.0093</td>
<td>0.0490</td>
</tr>
<tr>
<td>RNG</td>
<td>0.2777</td>
<td>0.0316</td>
<td>0.0411</td>
<td>0.0274</td>
<td>0.0263</td>
</tr>
<tr>
<td>MKW</td>
<td>0.2111</td>
<td>0.0463</td>
<td>0.0508</td>
<td>0.0161</td>
<td>0.0430</td>
</tr>
<tr>
<td>MKE</td>
<td>0.2106</td>
<td>0.0295</td>
<td>0.0382</td>
<td>0.0198</td>
<td>0.0608</td>
</tr>
</tbody>
</table>

Table 3.1 indicates that at $z/r_b = 4$, the HKW model produces the best prediction of radial velocity. However, this best prediction overpredicts the radial velocity by a factor of two close to the blade. Wilcox has shown that the HKW model fails to predict the sharp peak of kinetic energy near solid walls. This leads to an underprediction of the kinetic energy near the blade surface which can be observed at $z/r_b = 4$ in Fig. 3.8. This underprediction of kinetic energy causes an underprediction of eddy viscosity, which in turn causes the radial velocity to be overpredicted. There are three damping functions in the LKW model: $\beta^*$ in the $k$ equation, $\sigma$ in the $\omega$ equation, and $\sigma^*$ in the eddy viscosity formulation. The damping function $\beta^*$ is formulated to increase the kinetic energy near solid walls. At $z/r_b = 4$ in Fig. 3.8 the kinetic energy predictions by the LKW model are similar to the predictions by the HKW model. This indicates that $\beta^*$ is probably ill-suited for the complex flow in the impeller region since the kinetic energy is not increased near the blade surface. The damping function $\sigma$ is formulated to increase the dissipation near the wall so that the increased kinetic energy does not cause the eddy viscosity to become overpredicted. A similar purpose is served by $\sigma^*$, which also helps to prevent an overprediction of eddy viscosity near the wall. Turning attention back to Fig. 3.6, it
can be seen that the largest overprediction of radial velocity at $z/r_b = 4$ is produced by
the LKW model. This indicates that although the kinetic energy is similar to the HKW
model in this region, the eddy viscosity has been severely overdamped near the impeller
blade surface causing the increased overprediction of radial velocity. This suggests that
either $\sigma$ has caused the dissipation to be excessively overpredicted near the blade surface,
or $\sigma^*$ has overdamped the eddy viscosity near the blade surface. Similar arguments can
be made about the damping functions in other models. As stated above, the rms error
for the radial velocity prediction of the (HKW) model is low compared with the other
models, but the rms error in the kinetic energy for the (HKW) model is high compared
with the other models. This means that there must be significant error in the dissipation
rate computed by the (HKW) model. Experimental data for turbulence dissipation is
needed for a better assessment of the relationship between the turbulence predictions and
the velocity predictions. The combination of the ill-suited damping functions and the
inability for two-equation models to accurately predict anisotropic turbulence, causes poor
predictions of velocity and turbulence quantities in the impeller region. From Table 3.1, it

<table>
<thead>
<tr>
<th>Model</th>
<th>$u_r$</th>
<th>$u_z$</th>
<th>$u_\theta$</th>
<th>$k$</th>
<th>$k$ mean</th>
</tr>
</thead>
<tbody>
<tr>
<td>HKW</td>
<td>0.2906</td>
<td>0.0995</td>
<td>0.1114</td>
<td>0.0182</td>
<td>0.0208</td>
</tr>
<tr>
<td>LKW</td>
<td>0.3633</td>
<td>0.0829</td>
<td>0.1114</td>
<td>0.0141</td>
<td>0.0268</td>
</tr>
<tr>
<td>LKE</td>
<td>0.2559</td>
<td>0.0196</td>
<td>0.0923</td>
<td>0.0135</td>
<td>0.0460</td>
</tr>
<tr>
<td>RNG</td>
<td>0.1846</td>
<td>0.0318</td>
<td>0.1760</td>
<td>0.0190</td>
<td>0.0230</td>
</tr>
<tr>
<td>MKW</td>
<td>0.2046</td>
<td>0.0552</td>
<td>0.1078</td>
<td>0.0134</td>
<td>0.0455</td>
</tr>
<tr>
<td>MKE</td>
<td>0.1239</td>
<td>0.0332</td>
<td>0.1316</td>
<td>0.0268</td>
<td>0.0525</td>
</tr>
</tbody>
</table>

Table 3.2 Residual error (case 2, $z/r_b = 2.4$)

can be seen that the lowest rms error for $u_r$ and $k$ occurs for the HKW model while the
lowest rms error for $u_z$ and $u_\theta$ occurs for the MKE model. From Table 3.2, the MKE
model has the lowest error in $u_r$, while the LKE model has the lowest error in $u_z, u_\theta$, and $k$. Based on this simplistic measure of error, it appears that the LKE model provides the best description of the flow field.

Nonlinear models have allowed more accurate predictions of the turbulence quantities in non-isotropic regions of some flows without introducing any additional differential equations. Nonlinear models are usually numerically stiff compared with linear models, but would probably produce more accurate predictions of turbulent quantities in STRs, especially in the impeller discharge region.

Durbin [1] proposed a new elliptic relaxation model for the strongly inhomogeneous region near the wall in wall-bounded turbulent shear flows. The $k - \varepsilon$ models fail to predict near wall turbulence because $k^2/\varepsilon$ has the wrong profile as a function of $y^+$ near the wall. Durbin [1] suggests that the velocity scale near the wall is not $k$ but $\bar{u}^2$, where $\bar{u}^2$ is the variance of the normal component of turbulent velocity. In Durbin’s V2F model, the eddy viscosity is described as $\nu_t = C_{\mu} \bar{u}^2 T$ instead of as in the traditional $k - \varepsilon$ model formulation, and the model can be integrated all the way to the wall without any ad hoc damping functions. Verzicco [35] applied the V2F model to the STR of Dong. Their simulations show that the V2F model performs well near the shaft compared with the standard $k - \varepsilon$ model, but overpredicts the radial velocity in the impeller region. They also show that the turbulent kinetic energy is underpredicted in the impeller swept region. Thus, application of the $\bar{u}^2 f - k \varepsilon$ model also appears to result in the similar levels of inaccuracy as the other $k - \varepsilon$ models discussed in this chapter. There is therefore a need
for continued development of the two-equation models for complex flows.

3.3 Summary of Stirred Tank Results

The flow in an unbaffled stirred tank is investigated numerically using six different two-equation turbulence models. The mean velocity fields computed using the six models are compared with experimental LDV data. This is the first study on STRs which examines the performance of DNS-based low-Re $k - \varepsilon$ models and the performance of low-Re $k - \varepsilon$ models.

Specific observations made in this study are: 1) The radial velocity component in the impeller discharge region is overpredicted by each of the models. 2) The tangential velocity component in the impeller discharge region is predicted well by the models, but is underpredicted near the shaft. 3) The LKE model is the only model which produces reasonable kinetic energy predictions in the impeller discharge region.

Each model captures the qualitative circulation patterns in the STR. However, all of the models overpredict the mean radial discharge of the impeller due to an underprediction of the eddy viscosity. The experiments of Dong [22], show that the flow in this region is non-isotropic. To account for the anisotropy in the flow, more sophisticated turbulence models must be employed. The recent work at CTR Verzicco [35] suggest that more than one blade segment on the impeller must be modeled to capture the effects of neighboring blade vortex interactions.
Chapter 4. Elliptic Relaxation and the Development of a New $\overline{v^2 f}$ Model

4.1 Near Wall Velocity Scaling

In the $k - \varepsilon$ model, the eddy viscosity is computed as

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

(4.1)

where $C_\mu$ is typically 0.09. Figure 4.1 shows the computed value of $\nu_t$ for $C_\mu = 0.09$ and $C_\mu = 0.075$.

![Figure 4.1. Exact eddy viscosity for flow in a channel.](image)

It can be seen that $\nu_t$ is overpredicted for both values of $C_\mu$. This shows that Eq. 4.1 cannot accurately predict the near wall eddy viscosity as long as $C_\mu$ is a constant. This observation has motivated many researchers to develop functions which can be used to replace $C_\mu$. These functions are formulated in order to damp the eddy viscosity near the wall in order to prevent the overprediction observed in Fig. 4.1. Durbin [1] notes that the $k - \varepsilon$ model fails not because $C_\mu$ has the wrong value, but because $k^2/\varepsilon$ has the wrong
profile as a function of \( y^+ \). Based on this observation Durbin introduces an alternative eddy viscosity formulation defined as

\[
\nu_t = C \mu \frac{\overline{\nu^2 k}}{\varepsilon}
\]

where \( C = 0.2 \) and \( \overline{\nu^2} \) is the velocity fluctuation normal to the wall. Figure 4.1 shows \( \nu_t \) computed with the above expression and it is clear that this scaling produces the correct eddy viscosity in the near wall region. It seems at this point that if a scalar equation for \( \overline{\nu^2} \) can be derived which can be solved along with the \( k \) and \( \varepsilon \) equations, then the correct eddy viscosity can be computed near the wall without the need of damping functions. Durbin has developed a new model called the \( \nu^2 f - k \varepsilon \) model, which solves a scalar transport equation for \( k, \varepsilon, \) and \( \overline{\nu^2} \). The model is derived based on the idea of elliptic relaxation. At this point a summary of elliptic relaxation will be given, which follows from Durbin [1].

### 4.2 Elliptic Relaxation

Important issues in near wall modeling are boundary conditions and non-local wall effects. These non-local wall effects are elliptic in nature and are usually referred to as ‘pressure reflection’ or ‘pressure echo’. According to Durbin, these non-local wall effects originate from the surface boundary condition for the pressure fluctuation, \( p' \). The equation for \( p' \) is a Poisson equation, which can be derived from the Navier-Stokes equations.

\[
\nabla^2 p' = -\rho \left( \frac{\partial u_l \partial u_k}{\partial x_k \partial x_l} - \frac{\partial u_l \partial u_k}{\partial x_l \partial x_k} \right) - 2\rho \frac{\partial U_k \partial u_l}{\partial x_l \partial x_k}
\]

Equation 4.3 is usually characterized by dividing the equation into two parts. The first term on the right hand side is considered to be the slow part since it does not consist of any
mean velocity components. The second term does consist of mean velocity components so that it is considered to be the rapid part. In the near wall region mean velocity gradients are more dominant compared to the correlation of gradients of fluctuating quantities. Therefore, the slow part is neglected and Eq. 4.3 becomes

$$\nabla^2 p' = -2\rho \frac{\partial U_k}{\partial x_l} \frac{\partial u_l}{\partial x_k}$$

(4.4)

The solution to Eq. 4.4 is

$$p'(x) = \frac{1}{4\pi} \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} \frac{2\rho \partial_l U_k \partial'_k u_l (x')}{|x - x'|} d^3x'$$

(4.5)

in unbounded space. The wall boundary condition to Eq. 4.4 is \( \partial_y p' = \mu \partial^2_y v' \), where \( v' \) is the fluctuating velocity. Since \( \mu \partial^2_y v' \) is usually thought to be negligible, the boundary condition \( \partial_y p' = 0 \) is typically used. The solution to the Eq. 4.4 consists of a particular part, influenced by the right hand side, and a homogeneous part, influenced by the boundary condition. Since the boundary condition contributes to the solution interior to the fluid it is thought of as a non-local, kinematic effect. To solve Eq. 4.4 subjected to the boundary condition \( \partial_y p' = 0 \), the method of images can be used which adds a wall echo term to Eq. 4.5 to enforce the boundary condition. Equation 4.5 becomes

$$p'(x) = \frac{1}{2\pi} \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} \rho \partial_l U_k \partial'_k u_l (x', z', |y'|) \frac{d^3x'}{|x - x'|}$$

(4.6)

where we have absorbed the contribution of the image term by taking the absolute value of the wall normal direction. Instead of solving Eq. 4.6, Durbin [1] presents some rational for deriving a non-homogeneous, elliptic equation to represent the non-local, kinematic effects.
To explain the rationale, Eq. 4.5 is differentiated with respect to $x_i$ to get
\[
\frac{\partial}{\partial x_i} p(x) = \frac{\partial}{\partial x_i} \left[ \frac{1}{4\pi} \int \int \int_{-\infty}^{\infty} \frac{2 \rho \partial t U_k \partial'_{i} u_l (x')}{|x - x'|} d^3 x' \right]
\] (4.7)
Since the above integral is evaluated at $x'$ instead of $x$ the derivative can be taken inside of the integral so that Eq. 4.7 becomes
\[
\frac{\partial}{\partial x_i} p(x) = \frac{1}{2\pi} \int \int \int_{-\infty}^{\infty} \rho \partial t U_k \partial'_{i} u_l (x') \frac{1}{|x - x'|} d^3 x'
\] (4.8)
which contains the free-space Green function
\[
\frac{1}{4\pi |x - x'|}
\] (4.9)
The importance of this will become apparent shortly. To retain the free-space Green function the differential $\partial/\partial x_i$ is changed to $\partial/\partial x'_i$ and Eq. 4.8 becomes
\[
\frac{\partial}{\partial x'_i} p(x) = -\frac{1}{2\pi} \int \int \int_{-\infty}^{\infty} \rho \partial t U_k \partial'_{i} u_l (x') \frac{1}{|x - x'|} d^3 x'
\] (4.10)
Equation 4.10 can be integrated by parts. As a reminder the method of integration by parts states that for some function $u$,
\[
\int_{a}^{b} u dv = uv|_{a}^{b} - \int_{a}^{b} v du
\] (4.11)
where $uv$ is evaluated at the limits $a$ and $b$. To put Eq. 4.10 into the form of Eq. 4.11 let
\[
u = \rho \partial t U_k \partial'_{i} u_l (x')
\]
and
\[
dv = \frac{\partial}{\partial x'_i} \frac{1}{|x - x'|}
\]
Making these substitution, Eq. 4.11 becomes
\[
\frac{\partial}{\partial x'_i} p(x) = -\frac{1}{2\pi} \left[ \rho \partial t U_k \partial'_{i} u_l (x') \frac{1}{|x - x'|} - \int \int \int_{-\infty}^{\infty} \partial'_{i} \rho \partial t U_k \partial'_i u_l (x') \frac{1}{|x - x'|} d^3 x' \right]
\] (4.12)
where the first term in the brackets must be evaluated at the limits $-\infty$ and $\infty$, which causes the first term to become zero. Without the first term, Eq. 4.12 becomes

$$\partial_i p(x) = \rho \partial_i U_k \frac{1}{2\pi} \iint_{-\infty}^{\infty} \partial'_i \partial'_k u_l \left( x' \right) \frac{1}{|x - x'|} d^3 x'$$  \hspace{1cm} (4.13)$$

where the term $\rho \partial_i U_k$ has been taken out of the integrand because mean flow gradients are constant for homogeneous flow.

By rearranging the terms in Eq. 1.8 the second moment closure equation can be written as

$$\partial_t u_i u_j + U_k \partial_k u_i u_j = -\frac{1}{\rho} \left( u_j \partial_i p' + u_i \partial_j p' \right) - 2\nu \partial_k u_i \partial_k u_j - \partial_k u_k u_i u_j$$  \hspace{1cm} (4.14)$$

where

$$\frac{u_j \partial_i p + u_i \partial_j p}{\rho} = \rho \phi_{ij}$$  \hspace{1cm} (4.15)$$
is the velocity-pressure gradient correlation. Substituting the pressure gradients at $i$ and $j$ from Eq. 4.13 into Eq. 4.15, multiplied by $u_j$ and $u_i$ and then time averaging yields

$$\rho \phi_{ij} = \rho \partial_i U_k \frac{1}{2\pi} \iint \left[ \frac{u_j(x) \partial'_i \partial'_k u_l \left( x' \right) + u_i(x) \partial'_i \partial'_k u_l \left( x' \right)}{|x - x'|} \right] \frac{1}{|x - x'|} d^3 x'$$  \hspace{1cm} (4.16)$$

Note that time averaging only applies to turbulent fluctuating quantities. Also, notice that the right hand side contains two-point correlations which means that the second moment closure equations are unclosed and they depend on non-local effects. If the flow is homogeneous then Durbin shows that $\left[ \frac{u_j(x) \partial'_i \partial'_k u_l \left( x' \right) + u_i(x) \partial'_i \partial'_k u_l \left( x' \right)}{|x - x'|} \right]$ can be represented as the product of a fourth order constant tensor $M_{ijkl}$ and the mean velocity gradient.

$$-\rho \phi_{ij} = M_{ijkl} \partial_l U_k$$  \hspace{1cm} (4.17)$$
Both sides of Eq. 4.17 contain only single point correlations which can be computed. If the flow is non-homogeneous then the spatial correlation in the integrand must be represented. To do this, Durbin models \( \rho \partial_t U_k \left[ u_j(x) \partial_i \partial_k u_l(x') + u_i(x) \partial_i \partial_k u_l(x') \right] \) from Eq. 4.16 as

\[
\rho \partial_t U_k \left[ u_j(x) \partial_i \partial_k u_l(x') + u_i(x) \partial_i \partial_k u_l(x') \right] = Q_{ij} \left( x' \right) e^{-\frac{|x-x'|}{L}} \tag{4.18}
\]

where \( Q_{ij} \) represents a source term to be determined. Substituting Eq. 4.18 into Eq. 4.16 produces

\[
\int \int \int Q_{ij} \left( x' \right) e^{-\frac{|x-x'|}{L}} \frac{1}{4\pi |x-x'|} d^3x' \tag{4.19}
\]

According to Durbin the exponential represents the statistical decorrelation between distant eddies whereas the denominator represents the decay of the pressure field with distance from the point source. Equation 4.19 is the solution to a modified Helmholtz equation for \( \rho \phi_{ij} \).

\[
\nabla^2 \rho \phi_{ij} - \frac{\rho \phi_{ij}}{L^2} = -Q_{ij} \tag{4.20}
\]

This implies that a modified Helmholtz equation could be used to model non-local wall effects. The form of this equation chosen by Durbin is

\[
L^2 \nabla^2 f_{ij} - f_{ij} = -Q_{ij} \tag{4.21}
\]

where \( f_{ij} \) is an intermediate function. The point of the above derivation was to develop a method for modeling the scalar equation for the wall normal velocity fluctuation \( \overline{v^2} \). At this point we have developed a differential equation that can be solved, given the correct source term and boundary conditions, to represent the velocity-pressure gradient correlation in the second moment closure equation. Equation 4.14 can now be closed by using the solution.
of Eq. 4.21 for the pressure gradient terms of Eq. 4.15 and applying the standard closure approximations to the remaining terms. The normal fluctuating velocity $\vec{v}^2$ corresponds to $i = j = 2$ in Eq. 4.14. Therefore, the $f_{22}$ component of Eq. 4.21 is the only component that needs to be solved. This brings us to the new model developed by Durbin [1] which solves a scalar equation for $\vec{v}^2$, $k$, $\varepsilon$, and $f_{22}$. For simplicity, the function $f_{22}$ will hereafter simply be called $f$. In the above expressions the length scale $L$ is defined as

$$L = \max \left\{ C_L \frac{k^{3/2}}{\varepsilon}, C_\eta \left( \frac{\nu^3}{\varepsilon} \right)^{1/4} \right\}$$  \hspace{1cm} (4.22)

where $C_L$ and $C_\eta$ are constants. In Eq. 4.22 $L = C_L k^{3/2}/\varepsilon$ is chosen away from the wall, and near the wall $L = C_\eta (\nu^3/\varepsilon)^{1/4}$ is chosen so that the length scale $L$ does not become smaller than the Kolmogoroff scale.

4.2.1 The $\vec{v}^2 f - k \varepsilon$ Model

4.2.1.1 The Model equations

Durbin [1] incorporates the elliptic relaxation approach just described above to develop the $\vec{v}^2 f - k \varepsilon$ model. The transport equations for this model are

$$\frac{D\vec{v}^2}{Dt} = kf - \varepsilon \frac{\vec{v}^2}{k} + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_t) \frac{\partial \vec{v}^2}{\partial x_j} \right]$$ \hspace{1cm} (4.23)

$$L^2 \nabla^2 f - f = \frac{C_1}{T} \left( \frac{\vec{v}^2}{k} - \frac{2}{3} \right) + C_2 \frac{P}{k}$$ \hspace{1cm} (4.24)

$$\frac{Dk}{Dt} = P - \varepsilon + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$ \hspace{1cm} (4.25)

$$\frac{D\varepsilon}{Dt} = C_{\varepsilon_1} \frac{\varepsilon}{k} P - C_{\varepsilon_2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right]$$ \hspace{1cm} (4.26)

where $L$ is given by Eq. 4.22 and $T = \max \left[ \frac{k}{\varepsilon}, 6 \sqrt{\frac{\nu}{\varepsilon}} \right]$. The model coefficients are as follows:
\[ C_1 = 0.4, \quad C_2 = 0.3, \quad C_L = 0.3, \quad C_\eta = 70, \quad C_\varepsilon 2 = 1.9, \quad \sigma_\varepsilon = 1.0, \]
\[ C_\varepsilon 1 = 1.3 + 0.25/ \left[ 1 + (C_Ld/2L)^2 \right]^4 \]

The eddy viscosity is defined as \( \nu_t = C_\mu \overline{v^2} T \), where \( C_\mu = 0.2 \). The boundary condition for the \( f \)–equation is:
\[ f_w = -20\nu^2 \lim_{y \to 0} \frac{\frac{\overline{v^2}}{\varepsilon_w y^4}}{\varepsilon_w y^4} \]  

on a wall, \( y = 0 \), where \( \varepsilon_w \) is the value of \( \varepsilon \) at the wall. The \( \overline{v^2}f - k\varepsilon \) model has shown to produce better predictions for several flows when compared to the \( k - \varepsilon \) model.

### 4.2.1.2 Applications of the \( \overline{v^2}f - k\varepsilon \) Model

The original \( \overline{v^2}f - k\varepsilon \) model was formulated for wall bounded flows. More recently, the model has been tested for strongly separated flows, Durbin [36]. For wall bounded flows, \( \overline{v^2} \) represents the normal component of turbulence intensity, but for separated flows \( \overline{v^2} \) is simply a velocity scale which has surface boundary conditions suitable for the normal component of turbulence intensity near a wall. The model was tested for predicting the flow over a backward facing step of Driver [37] and the backward facing step of Jovic [38] for which \( k - \varepsilon \) models typically underpredict the reattachment location downstream of the step. This underprediction in the reattachment location is due to an overprediction in the eddy viscosity near the wall. The \( \overline{v^2}f - k\varepsilon \) model which predicts a reattachment location which is in good agreement with the experimentally determined reattachment location. The \( \overline{v^2} \) profile does not have the sharp peak near the wall that the kinetic energy profiles display which prevents the overprediction of eddy viscosity near the wall.

Durbin [39] has used the \( \overline{v^2}f - k\varepsilon \) model to predict the flow in the confined coaxial jet.
studied experimentally by Habib [40]. Durbin shows that the $\overline{v^2}f - k\varepsilon$ model correctly predicts the centerline velocity suggesting that the model predicts reasonably correct entrainment rates for the axisymmetric jet.

Standard turbulence models fail dramatically when predicting stagnating flows because they overpredict the kinetic energy and eddy viscosity at the stagnation point. This is usually referred to as the stagnation point anomaly and is caused from deficient representation of normal stress anisotropy by typical eddy viscosity formulas. Durbin [39] applied the $\overline{v^2}f - k\varepsilon$ model to the impinging jet flow of Cooper [41] and showed good agreement between predictions and experiment for the Stanton number.

4.2.1.3 Different Versions of the $\overline{v^2}f - k\varepsilon$ Model

There are three different versions of the $\overline{v^2}f - k\varepsilon$ model. Versions 1 and 2 differ by the formulation for $C_{\varepsilon 1}$ in the $\varepsilon$ equation. Version 2 was developed so that the coefficient $C_{\varepsilon 1}$ does not depend on the distance to the wall. Version 3 was developed so that the $f$ equation, Eq. 4.24, could be reformulated such that a trivial wall boundary condition, $f_w = 0$, could be used. This new boundary condition should improve the convergence of the model, although the authors have not verified this.

**Version 1:** Equations 4.23-4.26 are considered to be version 1. For this version the coefficient $C_{\varepsilon 1}$ is computed as:

$$C_{\varepsilon 1} = 1.3 + 0.25 \left[ \frac{1}{1 + (C_L d/2L)^2} \right]^4$$  \hspace{1cm} (4.28)

where $C_L = 0.3$, and $d$ represents the distance to the wall. The length scale $L$ is given in Eq. 4.22.
**Version 2** : To eliminate the need to compute the wall distance $d$, a new expression for $C_{\varepsilon 1}$ is used which uses the ratio $k/\overline{v^2}$. The new formulation for $C_{\varepsilon 1}$ and the new closure coefficients are:

$$C_{\varepsilon 1} = 1.4 \left( 1 + 0.045 \sqrt{k/\overline{v^2}} \right)$$

(4.29)

$$C_\mu = 0.22, \quad C_L = 0.25, \quad C_\eta = 85$$

where all other coefficients have remained the same.

**Version 3** : To create a more numerically friendly model, the $f$ equation has been reformulated so that instead of solving for $f$, a similar variable $\tilde{f}$ is solved. By adding an additional source term to the $f$ equation and subtracting a similar term from the $\overline{v^2}$ equation the new $\overline{v^2}$ and $\tilde{f}$ equations become:

$$\frac{D\overline{v^2}}{Dt} = kf - 6\varepsilon \frac{\overline{v^2}}{k} + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_t) \frac{\partial \overline{v^2}}{\partial x_j} \right]$$

$$L^2 \nabla^2 \tilde{f} - \tilde{f} = \frac{C_1}{T} \left( \frac{\overline{v^2}}{k} - \frac{2}{3} \right) + C_2 \frac{P}{k} + 5 \frac{\overline{v^2}}{kT}$$

and

$$C_{\varepsilon 1} = 1.4 \left( 1 + 0.045 \sqrt{k/\overline{v^2}} \right), \quad C_L = 0.23$$

The boundary condition for $\tilde{f}$ at the wall is $\tilde{f}_w = 0$.

Kalitzin [42] tested each version for predicting the flow over an airfoil. Version 1 was shown to produce better skin friction and pressure distribution results when compared to the other versions. He found that version 2 over-predicts the pressure on the suction surface of the blade and versions 2 and 3 overpredict the skin friction over the largest part of the wing. The differences in version 1 and 2 is contributed to the difference in the $C_{\varepsilon 1}$ approximations. The value of $C_{\varepsilon 1}$ for version ranges between 1.3 in the freestream to
1.55 near the wall whereas $C_{\varepsilon_1}$ varies between 1.54 in the freestream to 1.64 near the wall. Version 1 is also less numerically stiff because Eq. 4.28 is easier to control compared to Eq. 4.29.

4.2.1.4 **Drawbacks of the $\overline{v^2 f} - k\varepsilon$ Model**

Overall, the $\overline{v^2 f} - k\varepsilon$ model has been shown to produce better results compared with conventional two-equation turbulence model results for a wide range of flow applications. The model’s use of an equation for $\overline{v^2}$ introduces an anisotropic eddy viscosity formulation near solid walls. Although the $\overline{v^2 f} - k\varepsilon$ model has shown to provide improved predictions, there are some distinct drawbacks as discussed below. The model is known to be numerically stiff. The numerical stiffness is associated with the boundary condition in Eq. 4.27. The denominator $\varepsilon_w y^4$ can take on very small values in some situations causing the wall value of $f_w$ in Eq. 4.27 to take on large values resulting in numerical difficulties. For instance, for high Reynolds number flows the near wall grid spacing can become extremely small resulting in excessively small values of $y$. This can be seen in Fig. 4.2 where the grid has been cluster near solid walls in order to adequately resolve the the near wall velocity and turbulence. Also, the value of $\varepsilon_w$ can become extremely small as the flow moves around sharp corners. These small values of $\varepsilon_w$ are a result of the boundary condition $\varepsilon_w = 2v k / y^2$ applied to the $\varepsilon$-equation. Small values of $\varepsilon_w$ can cause $f_w$ to spike near the wall. Figure 4.3 shows contours of $\varepsilon$ for flow over a backward facing step. The results of the simulations for this problem will be discussed latter but Fig. 4.3 shows that the dissipation is high on the top wall of the step but becomes quite small along the
Computational Mesh for Low Reynolds Number Modeling

Figure 4.2. Computation Mesh

Turbulent Dissipation for the $\nu^2f-k\varepsilon$ model

$f_{22}$ for the $\nu^2f-k\varepsilon$ model

Figure 4.3. Near wall behavior of dissipation

vertical and lower walls. The low values are due to the fact that the kinetic energy is low since the velocity separates at the top wall creating a weak circulating flow near the vertical
and bottom walls. In addition to low $\varepsilon$ near the wall, the $\varepsilon$-equation cannot be solved all the way down to the wall because it becomes singular causing additional instabilities in the solution. To eliminated this problem typically the "damping functions" mentioned earlier are used or the $\varepsilon$ equation is formulated in terms of a time scale with appropriate bounds. This mathematically should fix the problem but since iterative methods are used to solve the equations, the solution is usually unphysical until convergence is achieved. Only at this point can it be expected that these remedies will be affective. This causes additional instabilities in the solution. In the near wall region the length scale $L$ can become extremely small. This can reduce the diffusion term of Eq. 4.24 enough to cause the $f$ equation to be influenced primarily by the source terms. Typical solvers perform best when used to solve equations which are dominated by convection and diffusion whereas source term dominated equations can be numerically stiff. This is primarily why spikes in the wall boundary condition for $f_w$ have such a large effect on the convergence of the model.

In fact, the common practice is to first solve the problem with a $k-\varepsilon$ model and then restart the solution using the $\overline{u^2}f - k\varepsilon$ model. This means the problem must be solved twice in order to get a converged $\overline{u^2}f - k\varepsilon$ solution. A typical convergence history of the $\overline{u^2}f - k\varepsilon$ model starting from an arbitrary initial guess is shown in Fig. 4.4. The poor convergence history of the $\overline{u^2}f - k\varepsilon$ model is evident in this figure. This is the motivation for the development of a new model which conserves the predictive accuracy of the $\overline{u^2}f - k\varepsilon$ model while improving it’s numerical stiffness.
4.2.2 The $\overline{v^2f} - k\omega$ Model

In the previous section we discussed how small values of $\varepsilon_{w}$ can cause the value for $f$ near walls to become extremely large causing numerical stiffness. In the $k - \omega$ model of Wilcox [11], the turbulent dissipation is approximated as $\varepsilon = \beta^{*}\omega k$ where $\beta^{*}$ is a closure coefficient. This approximation was formulated primarily based on dimensional arguments but has shown to predict accurate values of $\varepsilon$ for several flows. Since $\omega$ is infinite at a wall, this approximation for $\varepsilon$ always maintains a large value near the wall compared to experiments. For the $k - \omega$ model an equation for $\omega$ is solved instead of the $\varepsilon$-equation of the $k - \varepsilon$ model. A big advantage of the $\omega$ equation is that unlike the $\varepsilon$ equation, it can be solved all the way down to the wall. This is possible because the $\omega$ equation does not contain the singularities which are present in the $\varepsilon$ equation. The singularities in the $\varepsilon$ equation cause severe numerical stiffness for the $k - \varepsilon$ model and has received allot of attention by researchers trying to improve the convergence rate of the model. This suggests that this approximation for $\varepsilon$ would help to eliminate the numerical stiffness of the original $\overline{v^2f} - k\varepsilon$ model. As pointed out with Eq. 4.2 the use of $\overline{v^2}$ in the numerator of the eddy viscosity provides the correct near wall velocity scaling.

Also, since this new model will use the $k - \omega$ equation we would like to proceed with its derivation in a similar fashion as that of the $k - \omega$ model.

The derivation of the $k - \omega$ model starts by making the approximation $\varepsilon = \beta^{*}\omega k$. Substituting this expression into Eq. 4.1 produces the following eddy viscosity formulation $\nu_t = k/\omega$, which is used in the original $k - \omega$ model. For the new model we start with the
expression $\varepsilon = \beta^* \omega k^n \overline{v^2}^{1-n}$ and substitute this into Eq. 4.2 to produce the eddy viscosity formulation $\nu_t = C_\mu k^n \overline{v^2}^{1-n} / \omega$. This produces a model with the correct near wall scaling but without the numerical stiffness encountered in the $\overline{v^2} f - k \varepsilon$ model. The governing equations for the model are

$$\frac{D v^2}{D t} = k f - \varepsilon \frac{v^2}{k} + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_t) \frac{\partial v^2}{\partial x_j} \right]$$ (4.30)

$$L^2 \nabla^2 f - f = C_1 \left( \frac{v^2}{k} - \frac{2}{3} \right) + C_2 \frac{P}{k}$$ (4.31)

$$\frac{D k}{D t} = P - \varepsilon + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial k}{\partial x_j} \right]$$ (4.32)

$$\frac{D \omega}{D t} = \frac{\alpha}{k} \left( \frac{\omega^2}{k} P - \beta \omega^2 \left( \frac{v^2}{k} \right)^{1-n} \right) + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial \omega}{\partial x_j} \right]$$ (4.33)

where $\varepsilon = \beta^* \omega k^n \overline{v^2}^{1-n}$. For the $k - \omega$ model of Wilcox [11] the $\omega$ equation is

$$\frac{D \omega}{D t} = \frac{\alpha}{k} \left( \frac{\omega^2}{k} P - \beta \omega^2 \left( \frac{v^2}{k} \right)^{1-n} \right) + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial \omega}{\partial x_j} \right]$$ (4.34)

Comparing Eqs. 4.33 and 4.34 shows that the sink term, $\beta \omega^2$, has changed to $\beta \omega^2 \left( \frac{v^2}{k} \right)^{1-n}$. Equations 4.33 and 4.34 have both been derived from the standard $\varepsilon$ equation

$$\frac{D \varepsilon}{D t} = C_e \varepsilon \frac{C_2 \varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial \varepsilon}{\partial x_j} \right]$$ (4.35)

by replacing $\varepsilon$ in Eq. 4.35 with the approximation $\varepsilon = \beta^* \omega k$ for the $k - \omega$ model, or $\varepsilon = \beta^* \omega k^n \overline{v^2}^{1-n}$ for the $\overline{v^2} f - k \varepsilon$ model. The term $\left( \frac{v^2}{k} \right)^{-n}$ tends to reduce the sink term $\beta \omega^2 \left( \frac{v^2}{k} \right)^{1-n}$ in the near wall region causing Eq. 4.33 to produce larger values of $\omega$ compared to Eq. 4.34. To understand why this is important a brief description of $k - \omega$ model will now be presented.
4.2.2.1 Review of Low and High Reynolds Number $k - \omega$ Model

The governing equations for the $k - \varepsilon$ model are as follows:

$$\frac{Dk}{Dt} = P - \varepsilon + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_t) \frac{\partial k}{\partial x_j} \right]$$

$$\frac{D\varepsilon}{Dt} = C_{\varepsilon 1} \frac{\varepsilon}{k} P - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right]$$

$$\nu_t = C_{\mu} k^2 / \varepsilon$$ (4.37)

The details of the coefficients of the above equations where presented in Chapter 2. These equations overpredict the near wall eddy viscosity when applied without the use of wall functions or low Reynolds number "damping functions". For the $k - \omega$ model this is not the case, because instead of solving a transport equation for $\varepsilon$ the approximation is made that $\varepsilon = \beta^* \omega k$. This approximation always overpredicts $\varepsilon$ near the wall because the wall boundary condition for $\omega$ tends to infinity. When this approximation is substituted into Eq. ?? and 4.37 the following equations are formed.

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

$$\frac{D\omega}{Dt} = \frac{\omega}{k} \left[ \alpha \frac{\omega^2}{k} P - \beta^* \omega^2 \right] + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right]$$

$$\nu_t = k / \omega$$ (4.40)

Once again a description of the coefficients of the $k - \omega$ model are presented in Chapter 2. When these equations are applied near the wall the kinetic energy is grossly underpredicted compared to experiments, but the eddy viscosity is predicted correctly. In order for the $k - \omega$ model to produce accurate near wall predictions of kinetic energy and eddy viscosity, low Reynolds number "damping functions" are required. In fact, there are three
"damping functions" needed, one in the kinetic energy equation, one in the equation for \( \omega \), and one in the eddy viscosity formulation. With these "damping functions" Eqs. 4.38-4.40 become:

\[
\frac{Dk}{Dt} = P - f_k \beta^* \omega k + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial k}{\partial x_j} \right] \\
\frac{D\omega}{Dt} = \alpha \frac{\omega}{k} P - f_\omega \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial \omega}{\partial x_j} \right] \\
\nu_t = f_\mu k/\omega
\] (4.41)

where the "damping functions" take on the following form.

\[
f_k = (0.025 + R_t/6) (1 + R_t/6)^{-1} \quad (4.43)
\]

\[
f_\mu = \left[ 0.278 + (R_t/8)^4 \right] \left[ 1 + (R_t/8)^4 \right]^{-1} \quad (4.44)
\]

\[
f_\omega = (0.1 + R_t/2.7) \left[ (1 + R_t/2.7) f_\mu \right]^{-1}
\]

Since the approximation \( \varepsilon = \beta^* \omega k \) overpredicts the dissipation, this in turn causes the kinetic energy to be underpredicted near the wall. The function \( f_k \) is formulated so that it damps this overprediction of dissipation so that the near wall kinetic energy is in better agreement with experiments. Since the eddy viscosity is predicted correctly when the kinetic energy is underpredicted, increasing the kinetic energy near the wall cause the eddy viscosity to be overpredicted. This is countered by the use of the function \( f_\mu \) in Eq. 4.42 which reduces the eddy viscosity near the wall as the kinetic energy is increased to match the experiments. In addition to damping the eddy viscosity the function \( f_\omega \) is used in the \( \omega \) equation which reduces the sink term causing an increase in the production of \( \omega \) near the wall. As one might expect, these near wall corrections have been derived for
simple flows, such as flow in a channel, and are typically ill suited for complex flows.

For the new $\overline{v^2}f - k\omega$ model we use the expression $\varepsilon = \beta^\ast\omega k^n\overline{v^2}^{1-n}$. This expression for epsilon also overpredicts the dissipation near the wall, which helps with numerical stability, but not to the extent as the original expression $\varepsilon = \beta\omega k$. This is the case because $\overline{v^2}$ is much smaller than $k$ near the wall and decays to zero much more rapidly as the wall is approached. Also, when the expression $\varepsilon = \beta^\ast\omega k^n\overline{v^2}^{1-n}$ is used to derive the new $\omega$ equation, Eq. 4.33, the sink term has changed from it’s original form, $\beta\omega^2$, to the new form, $\beta\omega^2\left(\frac{\overline{v^2}}{k}\right)^{1-n}$. Since $\overline{v^2}$ decays more rapidly as the wall is approached compared with $k$, this reduces the sink term causing an increase in the production $\omega$. This allows the $\overline{v^2}f - k\omega$ model to compute accurately predict the near wall kinetic energy and eddy viscosity without the need for "damping functions".

4.2.2.2 The $\overline{v^2}f - k\omega$ Model Coefficients

The model coefficients for the original $k - \omega$ model have the following values.

$$\beta^\ast = 0.09, \quad \alpha = 5/9, \quad \beta = 3/40, \quad \sigma_\omega = \sigma_k = 2.0$$

Since the new $\omega$ equation has changed from the original form of the $k - \omega$ model the coefficients must be rederived and a detailed explanation of this process is described here. The model coefficients can be determined by analyzing the mean-momentum equation and the equations for $k$ and $\omega$ in the log layer where these reduce to

$$0 = \frac{\partial}{\partial y} \left[ v_t \frac{\partial U}{\partial y} \right]$$

(4.45)

$$0 = v_t \left( \frac{\partial U}{\partial y} \right)^2 - \beta^\ast\omega k^n\overline{v^2}^{1-n} + \sigma_k \frac{\partial}{\partial y} \left[ v_t \frac{\partial k}{\partial y} \right]$$

(4.46)
\[ 0 = \frac{\alpha}{k} \omega \left( \frac{\partial U}{\partial y} \right)^2 - \beta \omega^2 \left( \frac{v^2}{k} \right)^{1-n} + \sigma \omega \frac{\partial}{\partial y} \left[ v_t \frac{\partial \omega}{\partial y} \right] \] (4.47)

Equation 4.45 can be integrated twice to obtain

\[ U = \frac{u_\tau}{\kappa} \ln y + C \] (4.48)

where \( C \) is a constant. A common approximation in the log layer is that the turbulent production rate is balanced by the rate of dissipation. For the \( \overline{v'f} - k\omega \) this translates into

\[ v_t \left( \frac{\partial U}{\partial y} \right)^2 = \beta^* \omega k^n \overline{v^2}^{1-n} \] (4.49)

According to Townsend [43], the ratio \( \overline{v^2}/k = 0.36 \) in the log layer. Making this substitution into Eq. 4.49 produces

\[ v_t \left( \frac{\partial U}{\partial y} \right)^2 = \beta^* \omega k (0.36)^{1-n} \] (4.50)

The eddy viscosity varies linearly with distance from the surface and is approximated as

\[ v_t = \kappa u_\tau y \] (4.51)

where \( \kappa = 0.41 \) and \( u_\tau \) is the friction velocity. Substituting Eq. 4.51 into Eq. 4.50 and solving for the product \( \omega k \), Eq. 4.50 becomes

\[ \omega k = \frac{u_\tau^3}{(0.36)^{1-n} \beta^* \kappa y} \] (4.52)

where \( \beta^* \) is a closure coefficient to be determined. If the approximation \( v_t = C \mu \frac{k^2}{\varepsilon} \) is reformulated with the new formulation for \( \varepsilon \), and the new expression is solved for \( \omega \), then the following expression is obtained

\[ \omega = \frac{k}{(0.36)^{1-n} \kappa u_\tau y} \] (4.53)
Plugging Eq. 4.53 back into Eq. 4.52 produces an expression for the kinetic energy.

\[ k = \frac{u_\tau}{\sqrt{C_\mu}} \]  
\( (4.54) \)

Plugging Eq. 4.54 back into Eq. 4.52 an expression for \( \omega \) can be obtained.

\[ \omega = \frac{\sqrt{C_\mu u_\tau}}{(0.36)^{1-n} \beta^* \kappa y} \]  
\( (4.55) \)

Substituting Eq. 4.51, Eq. 4.54, and Eq. 4.55 into Eq. 4.47 produces an expression for the closure coefficient \( \alpha \).

\[ \alpha = \frac{\beta}{\beta^*} - \sigma \left( \frac{\kappa^2}{\sqrt{C_\mu}} \right) \]  
\( (4.56) \)

According to Townsend [43] in the log layer \( u_\tau \) is equal to the Reynolds shear stress, \( \tau_{xy} \), and is constant. He also shows that \( \tau_{xy}/k \approx 0.3 \). From Eq. 4.54 \( u_\tau/k = \sqrt{C_\mu} \) which implies that \( C_\mu = 0.09 \). The ratio of \( \beta^* \) to \( \beta \) can be established by applying the model to decaying homogeneous, isotropic turbulence. For this kind of turbulence, there are no mean gradients of flow properties. Therefore, Eq. 4.32 and Eq. 4.33 become

\[ \frac{dk}{dt} = -\beta^* \omega k^n \frac{v^2}{k^{1-n}} \]  
\( (4.57) \)

\[ \frac{d\omega}{dt} = -\beta \omega^2 \left( \frac{\tau}{k} \right)^{1-n} \]  
\( (4.58) \)

Substituting the approximation \( \frac{v^2}{k} = 0.36k \) into Eq. 4.57 and Eq. 4.58 the following equations are formed

\[ \frac{dk}{dt} = -\beta^* \omega k^n \left( 0.36k \right)^{1-n} \]  
\( (4.59) \)

\[ \frac{d\omega}{dt} = -\beta \omega^2 \left( 0.36 \right)^{1-n} \]  
\( (4.60) \)

Integrating Eq. 4.60 and substituting this solution into Eq. 4.59 the asymptotic solution
becomes

\[ k \sim t^{-\beta^*/\beta} \]

Townsend [43] indicates that \( k \sim t^{-n} \) where \( n = 1.25 \pm 0.06 \). Setting \( \beta^*/\beta = 5/4 \) places the value in the center of the range. For the standard \( k - \omega \) model of Wilcox [11] Eq. 4.56 becomes

\[ \alpha = \frac{\beta}{\beta^*} - \frac{1}{\sigma_\omega} \left( \frac{\kappa_2^2}{\beta^*} \right) \]  (4.61)

Equation 4.56 and Eq. 4.61 are similar with the difference that \( C_\mu \) has replaced \( \beta^* \) in the second term. Comparing these two equations shows that \( C_\mu = \beta^* \) for the equations to have the same form. Now the value of \( \beta \) can be computed from the expression \( \beta^*/\beta = 5/4 \) which was stated above. The derivation of \( \sigma_k \) and \( \sigma_\omega \) are not quite as clear. Since in the \( k \) equation for the \( \overline{v^2}f - k\varepsilon \) model a value of \( \sigma_k = 1.0 \) is used this value was chosen for the \( \overline{v^2}f - k\omega \). Using a value of \( \sigma_\omega = 1.5 \) has shown to produce satisfactory results for both wall bounded and free shear flows. Table 4.1 shows a summary of the coefficients for original new \( \overline{v^2}f - k\omega \) model compared to those of the original \( k - \omega \) model. The main motivation for developing the \( \overline{v^2}f - k\omega \) model was to develop a model which is numerically stable while retaining the predictive capabilities of the \( \overline{v^2}f - k\varepsilon \) model. The \( \overline{v^2}f - k\varepsilon \) model is numerically stiff when starting the solution from an arbitrary initial

<table>
<thead>
<tr>
<th>( k - \omega )</th>
<th>( \overline{v^2}f - k\omega )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha )</td>
<td>0.5555</td>
</tr>
<tr>
<td>( \beta^* )</td>
<td>0.09</td>
</tr>
<tr>
<td>( \beta )</td>
<td>3/40</td>
</tr>
<tr>
<td>( \sigma_k )</td>
<td>2.0</td>
</tr>
<tr>
<td>( \sigma_\omega )</td>
<td>2.0</td>
</tr>
</tbody>
</table>
guess. In order to obtain a converged solution with the $v^2f - k\varepsilon$ model, the problem should be solved first using a $k - \varepsilon$ model in order to obtain reasonable values of $k$ and $\varepsilon$ before the $v^2f - k\varepsilon$ model is started. The convergence history for both the $v^2f - k\omega$ and $v^2f - k\varepsilon$ models is shown in Fig. 4.4. The simulations for both models were run without the help of an initially converged solution. Figure 4.4 shows that the $v^2f - k\varepsilon$ model will not run without an initially converged solution but will instead diverge and ultimately blow up.

![Convergence History](image)

**Figure 4.4. Convergence history**
5.1 General Transport Equations

The presentation of the governing equations will be started by a review of some of the notation. In this chapter we introduce a generic transport equation using several notations. The most general form is coordinate system independent and is often referred to in the literature as vector notation. We then write the equations in a Cartesian frame of reference. This single equation for a scalar variable is then generalized to describe a set of coupled transport equations for a vector of dependent variables. An arbitrary quantity \( \phi \) obeys a generalized conservation principle written as a differential equation of the form

\[
\frac{\partial}{\partial t} \left( \rho \phi \right) + \nabla \cdot \left( \rho u \phi \right) = \nabla \cdot \left( \Gamma \nabla \phi \right) + S
\]  

(5.1)

Terms in this equation include: the unsteady term, the convective term, a diffusion term and a general source term. The quantity \( \phi \) can represent any number of quantities including mass fraction of a chemical species, temperature, a velocity component, turbulent kinetic energy and turbulent dissipation rate. The exact form of the source term, \( S \) and the diffusion coefficient \( \Gamma \) depends on the scalar quantity \( \phi \). In Eq. 5.1, the divergence and gradient operators can be expanded in a Cartesian coordinate frame of reference with coordinate axes denoted as \( x, y, \) and \( z \) and respective velocity components \( u, v, \) and \( w \). The resulting identical mathematical expression becomes:

\[
\frac{\partial}{\partial t} \left( \rho \phi \right) + \frac{\partial}{\partial x} \left( \rho u \phi \right) + \frac{\partial}{\partial y} \left( \rho v \phi \right) + \frac{\partial}{\partial z} \left( \rho w \phi \right) = \frac{\partial}{\partial x} \left( \Gamma \frac{\partial \phi}{\partial x} \right) + \frac{\partial}{\partial y} \left( \Gamma \frac{\partial \phi}{\partial y} \right) + \frac{\partial}{\partial z} \left( \Gamma \frac{\partial \phi}{\partial z} \right) + S
\]  

(5.2)

Rearrangement of terms containing like behavior results in an expression with a more
compact form

\[ \frac{\partial}{\partial t} (\rho \phi) + \frac{\partial}{\partial x} \left( \rho u \phi - \Gamma \frac{\partial \phi}{\partial x} \right) + \frac{\partial}{\partial y} \left( \rho v \phi - \Gamma \frac{\partial \phi}{\partial y} \right) + \frac{\partial}{\partial z} \left( \rho w \phi - \Gamma \frac{\partial \phi}{\partial z} \right) = S \] (5.3)

Equation 5.3 can be written more compactly using indicial notation

\[ \frac{\partial}{\partial t} (\rho \phi) + \frac{\partial}{\partial x_j} \left( \rho u_j \phi - \Gamma \frac{\partial \phi}{\partial x_j} \right) = S \] (5.4)

where like indices in a term represent a summation over all possible values of the index.

The possible indicial values here are the number of spatial dimensions in the flow. The three spatial directions \( x, y, \) and \( z \) correspond to their respective velocity components \( u, v, \) and \( w. \)

We could also write Eq. 5.3 in terms of fluxes (to be described in the next section).

\[ \frac{\partial}{\partial t} Q + \frac{\partial}{\partial x} (E - E_v) + \frac{\partial}{\partial y} (F - F_v) + \frac{\partial}{\partial z} (G - G_v) = S \] (5.5)

where definitions of the conserved variable, \( Q, \) and scalar fluxes, \( E, F, \) and \( G \) (a \( \nu \) subscript is used to denote viscous and diffusive parts) are discernible by comparing Eq. 5.5 with Eq. 5.3. In the next section the governing equations will start with a form similar to Eq. 5.5; however, the turbulence transport equations will be presented in a form similar to Eq. 5.4.

5.2 Governing Equations

The Favre-averaged Navier-Stokes and turbulence model equations are described in this section. The reduced notation in the previous section will be used in this section; however, the notation for partial derivatives will be shortened. For example: \( \partial/\partial x \rightarrow \partial_x, \) etc.

The \( i\)th species partial pressure, density, mass fraction, molecular weight and enthalpy is
\( p_i, \rho_i, Y_i, h_i \) and the thermodynamic pressure, mixture density, velocity components and temperature are given by \( p, \rho, u, v, w, \) and \( T, \) respectively.

All species are assumed to obey an equation of state: \( p_i = \rho_i RT/W_i. \) The equations are solved simultaneously in a standard generalized frame of reference with coordinate directions denoted by \( \xi, \eta, \) and \( \zeta. \) The appropriate vector equation is of the form:

\[
\partial_t \hat{Q} + \partial_\xi \left( \hat{E} - \hat{E}_v \right) + + \partial_\eta \left( \hat{F} - \hat{F}_v \right) + \partial_\zeta \left( \hat{G} - \hat{G}_v \right) = \hat{H}
\] (5.6)

The hats are used to represent the flux vectors, \( \hat{E}, \hat{E}_v, \) etc. in the generalized coordinate system and are related to the flux vectors in the Cartesian \((x, y, z)\) frame:

\[
\hat{Q} = \frac{1}{J} Q, \quad \hat{H}_g = \frac{1}{J} H_g
\]
\[
\hat{E} = \frac{1}{J} \left( \xi_x E + \xi_y F + \xi_z G \right), \quad \hat{E}_v = \frac{1}{J} \left( \xi_x E_v + \xi_y F_v + \xi_z G_v \right)
\]
\[
\hat{F} = \frac{1}{J} \left( \eta_x E + \eta_y F + \eta_z G \right), \quad \hat{F}_v = \frac{1}{J} \left( \eta_x E_v + \eta_y F_v + \eta_z G_v \right)
\]
\[
\hat{G} = \frac{1}{J} \left( \zeta_x E + \zeta_y F + \zeta_z G \right), \quad \hat{G}_v = \frac{1}{J} \left( \zeta_x E_v + \zeta_y F_v + \zeta_z G_v \right)
\]

where \( \xi, \eta, \) and \( \zeta \) are spatial coordinates in the generalized frame of reference.

### 5.2.1 Inviscid Fluxes

The conserved variable vector and Cartesian inviscid fluxes are

\[
Q = [\rho Y_1, ..., \rho Y_N, \rho u, \rho v, \rho w, E_t]^T
\]
\[
E = [\rho u Y_1, ..., \rho u Y_N, \rho u^2 + p, \rho uv, \rho uw, (E_t + p) u]^T
\] (5.7)
\[
F = [\rho v Y_1, ..., \rho v Y_N, \rho uv, \rho v^2 + p, \rho vw, (E_t + p) v]^T
\]
\[
G = [\rho w Y_1, ..., \rho w Y_N, \rho vw, \rho ww, \rho w^2 + p, (E_t + p) w]^T
\]

The quantities \( \rho, p, u, v, w, \) and \( Y_s \) denote density, pressure, Cartesian velocity
components, and species mass fraction, respectively. The total energy is \( E_t = \rho \left[ e + \frac{1}{2} (u^2 + v^2 + w^2) \right] \), where \( e \) is the specific internal energy (the subscript \( T \) denotes conventional vector transpose).

### 5.2.2 Viscous Fluxes

The viscous fluxes are written as

\[
E_v = \begin{bmatrix} q_{x1}, \ldots, q_{xN}, \tau_{xx}, \tau_{xy}, \tau_{xz}, u\tau_{xx} + v\tau_{xy} + w\tau_{xz} + q_{xe} \end{bmatrix}^T
\]

\[
F_v = \begin{bmatrix} q_{y1}, \ldots, q_{yN}, \tau_{yx}, \tau_{yy}, \tau_{yz}, u\tau_{yx} + v\tau_{yy} + w\tau_{yz} + q_{ye} \end{bmatrix}^T
\]

\[
G_v = \begin{bmatrix} q_{z1}, \ldots, q_{zN}, \tau_{zx}, \tau_{zy}, \tau_{zz}, u\tau_{zx} + v\tau_{zy} + w\tau_{zz} + q_{ze} \end{bmatrix}^T
\]

with viscous stress components

\[
\tau_{xx} = 2\mu_e \partial_x u - \frac{2}{3} \mu_e (\partial_x u + \partial_y v + \partial_z w), \quad \tau_{xy} = \tau_{yx} = \mu_e (\partial_x u - \partial_y v)
\]

\[
\tau_{yy} = 2\mu_e \partial_y v - \frac{2}{3} \mu_e (\partial_x u + \partial_y v + \partial_z w), \quad \tau_{xz} = \tau_{zx} = \mu_e (\partial_z w - \partial_y w)
\]

\[
\tau_{zz} = 2\mu_e \partial_z w - \frac{2}{3} \mu_e (\partial_x u + \partial_y v + \partial_z w), \quad \tau_{yz} = \tau_{zy} = \mu_e (\partial_z v - \partial_y w)
\]

**Energy Fluxes**

The energy fluxes in the three coordinate directions are given by

\[
q_{xe} = k_e \partial_x T + \rho \sum_{s=1}^{N} h_s D_{s_m} \partial_x Y_s
\]

\[
q_{ye} = k_e \partial_y T + \rho \sum_{s=1}^{N} h_s D_{s_m} \partial_y Y_s
\]

\[
q_{ze} = k_e \partial_z T + \rho \sum_{s=1}^{N} h_s D_{s_m} \partial_z Y_s
\]

where \( T \) is temperature and \( k_e = k_l + C_p m \mu_t / Pr_t \) is the effective thermal conductivity with molecular conductivity \( k_l \) and turbulent Prandtl number \( Pr_t \).
5.2.3 Preconditioning

Low Mach number preconditioning effectively re-scales the acoustic scales to match that of the convective scales, Weiss [44]. A pseudo-time derivative of the dependent variable vector is added to the transport equations as follows

\[ \Gamma \partial_t \hat{U} + \partial_t \hat{Q} + \partial_\xi \left( \hat{E} - \hat{E}_v \right) + \partial_\eta \left( \hat{F} - \hat{F}_v \right) + \partial_\zeta \left( \hat{G} - \hat{G}_v \right) = \hat{H} \]  

The dependent variables and preconditioning matrix are defined by

\[ \hat{U} = \frac{1}{J} \left[ p_1, p_2, \ldots, p_N, u, v, w, T \right]^T \]

\[ \Gamma = \begin{bmatrix}
\frac{W_1}{RT} + \Theta Y_1 & \Theta Y_1 & \ldots & \Theta Y_1 & 0 & 0 & 0 & -\rho \\
\Theta Y_2 & \frac{W_2}{RT} + \Theta Y_2 & \ldots & \Theta Y_2 & 0 & 0 & 0 & -\rho \\
\ldots & \ldots & \ldots & \ldots & \ldots & \ldots & \ldots & \ldots \\
\Theta Y_N & \Theta Y_N & \ldots & \frac{W_N}{RT} + \Theta Y_N & 0 & 0 & 0 & -\rho \\
u \left( \Theta + \frac{W_1}{RT} \right) & u \left( \Theta + \frac{W_2}{RT} \right) & \ldots & u \left( \Theta + \frac{W_N}{RT} \right) & \rho & 0 & 0 & -\rho \\
v \left( \Theta + \frac{W_1}{RT} \right) & v \left( \Theta + \frac{W_2}{RT} \right) & \ldots & v \left( \Theta + \frac{W_N}{RT} \right) & 0 & \rho & 0 & -\rho \\
w \left( \Theta + \frac{W_1}{RT} \right) & w \left( \Theta + \frac{W_2}{RT} \right) & \ldots & w \left( \Theta + \frac{W_N}{RT} \right) & 0 & 0 & \rho & -\rho \\
\alpha_1 & \alpha_2 & \ldots & \alpha_N & \rho u & \rho v & \rho w & \rho \left( C_{pm} - \frac{H}{T} \right)
\end{bmatrix} \]

with

\[ U_{ref} = \min \left[ a^2, \max \left( \left| \vec{V} \right|^2, K \left| \vec{V}_\infty \right|^2 \right) \right] \]

\[ \Theta = \frac{1}{U_{ref}^2} - \frac{1}{a^2} \]

\[ \alpha_s = H \left( \Theta + \frac{W_s}{RT} \right) - 1 \]

\( H \) is enthalpy per unit mass, \( \left| \vec{V} \right| \) is the local velocity magnitude, \( \left| \vec{V}_\infty \right| \) is a reference velocity, \( a \) the sound speed and \( K \) is a constant (a value of unity is typically used).

5.2.4 Numerical Method

The \( \xi, \eta, \) and \( \zeta \) directions are discretized using \( j, k, \) and \( l, \) respectively. Second order backward 3-point physical time differencing is used and Euler differencing is used for the pseudo-variable, \( \tau. \) A second-order low diffusion flux-splitting scheme of Edwards [45] is
used for the convective terms and second order central differences are used for the viscous
terms. Choosing \( p \) as the pseudo-time index and \( n \) as the physical time index ( \( p_+ \) implies
\( p + 1 \) ), the discretized form of Eq. 5.8 becomes

\[
\frac{\Gamma}{\Delta \tau} \left( \tilde{U}_i^{p_+,n_+} - \tilde{U}_i^{p,n_+} \right) + \frac{1}{2 \Delta t} \left[ 3 \tilde{Q}_i^{p_+,n_+} - 4 \tilde{Q}_i^n + \tilde{Q}_i^{n-} \right] \\
+ \left[ \tilde{E} - \tilde{E}_u \right]_{j_+}^{p_+,n_+} - \left[ \tilde{E} - \tilde{E}_u \right]_{j_-}^{p_+,n_+} \\
+ \left[ \tilde{F} - \tilde{F}_v \right]_{k_+}^{p_+,n_+} - \left[ \tilde{F} - \tilde{F}_v \right]_{k_-}^{p_+,n_+} \\
+ \left[ \tilde{G} - \tilde{G}_u \right]_{l_+}^{p_+,n_+} - \left[ \tilde{G} - \tilde{G}_u \right]_{l_-}^{p_+,n_+} \\
= \tilde{H}_i^{p_+,n_+}
\]

The solution vector is the change in the dependent variable vector and is defined as:

\[
\delta \tilde{U}_i^{p_+} = \tilde{U}_i^{p_+} - \tilde{U}_i^p.
\]

Denoting \( \tilde{P}_{i\pm} \) as a generic numerical approximation to the fluxes \( \tilde{E}, \tilde{F}, \tilde{G} \) for \( i = j, k \) and \( l \) respectively, the linearizations for the inviscid flux, viscous flux,
conserved variable and source term vectors, respectively are:

\[
\begin{align*}
\tilde{P}_{i\pm}^{p_+,n_+} &= \tilde{P}_{i\pm}^{p,n_+} + \partial_{\tilde{U}} \tilde{P}_{i\pm} \delta \tilde{U}_i^{p_+,n_+} \\
\left( \tilde{P}_v \right)_{i\pm}^{p_+,n_+} &= \left( \tilde{P}_v \right)_{i\pm}^{p,n_+} + \partial_{\tilde{U}} \left( \tilde{P}_v \right)_{i\pm} \delta \tilde{U}_i^{p_+,n_+} \\
\tilde{Q}_i^{p_+,n_+} &= \tilde{Q}_i^{p,n_+} + \partial_{\tilde{U}} \tilde{Q}_i \delta \tilde{U}_i^{p_+,n_+} \\
\tilde{H}_i^{p_+,n_+} &= \tilde{H}_i^{p,n_+} + \partial_{\tilde{U}} \tilde{H}_i \delta \tilde{U}_i^{p_+,n_+}
\end{align*}
\] (5.9)

where \( \partial_{\tilde{U}} \tilde{P}, \partial_{\tilde{U}} \left( \tilde{P}_v \right), \partial_{\tilde{U}} \tilde{Q} \) and \( \partial_{\tilde{U}} \tilde{H} \) are the Jacobian matrices. Using the linearization
Eq. 5.9, the matrix equation for \( \delta \tilde{U} \) at the \( p_+ \) pseudo-time iteration of the \( n_+ \) physical time
level is given as:

\[
\left( \frac{\Gamma}{\Delta \tau} + \partial_\tau \tilde{H}_i + \frac{1.5}{\Delta t} \partial_\tau \tilde{Q}_i \right) \delta \tilde{U}^p,n+ \ni
+ \left[ \partial_\tau \tilde{E} - \partial_\tau \tilde{E}_v \right]_{j+} \delta \tilde{U}^p,n+ \ni
- \left[ \partial_\tau \tilde{E} - \partial_\tau \tilde{E}_v \right]_{j-} \delta \tilde{U}^p,n+ \ni
+ \left[ \partial_\tau \tilde{F} - \partial_\tau \tilde{F}_v \right]_{k+} \delta \tilde{U}^p,n+ \ni
- \left[ \partial_\tau \tilde{F} - \partial_\tau \tilde{F}_v \right]_{k-} \delta \tilde{U}^p,n+ \ni
+ \left[ \partial_\tau \tilde{G} - \partial_\tau \tilde{G}_v \right]_{l+} \delta \tilde{U}^p,n+ \ni
- \left[ \partial_\tau \tilde{G} - \partial_\tau \tilde{G}_v \right]_{l-} \delta \tilde{U}^p,n+ \ni
\]

\( = RHS^{p,n+} \)

where

\[
RHS^{p,n+} = - \left\{ \begin{array}{c}
\left[ 1.5 \tilde{Q}^{p,n+}_i - 2 \tilde{Q}^n_i + 0.5 \tilde{Q}^{n-}_i \right] / \Delta t \\
+ \left[ \tilde{E} - \tilde{E}_v \right]^{p,n+}_j - \left[ \tilde{E} - \tilde{E}_v \right]^{p,n+}_j \\
+ \left[ \tilde{F} - \tilde{F}_v \right]^{j,n+}_k - \left[ \tilde{F} - \tilde{F}_v \right]^{j,n+}_k \\
+ \left[ \tilde{G} - \tilde{G}_v \right]^{p,n+}_l - \left[ \tilde{G} - \tilde{G}_v \right]^{p,n+}_l + \tilde{H}^{p,n+}_i \end{array} \right\}
\]

Equation 5.10 can be solved using either a multi-level line relaxation scheme, ILU scheme or a full GMRES algorithm. The multi-level scheme involves solving the system on increasingly finer grid, interpolating the solution from the coarser to the finer grid as the solution proceeds. To take full advantage of the multi-leveling features, the number of grid cells in each coordinate direction must be divisible by 4 and should be a minimum of 12 cells. For example, a grid zone of dimensions 12 \times 12 cells would have a medium and coarse grid dimensions of 6 \times 6 and 3 \times 3 respectively.
Chapter 6. Validation of the $\bar{v}^2 f - k\omega$ Model

In this chapter the $\bar{v}^2 f - k\omega$ model was compared with the $\bar{v}^2 f - k\varepsilon$ model and some of the conventional two equation models for computing various flow problems. The problems in this chapter consist of flow in a two dimensional channel, flow over a backward facing step, flow in a heated cavity, and flow in coaxial jet. These problems have been chosen because there is reliable data for each flow and they represent standard benchmark problems that are typically used when testing turbulence models. The results are presented and compared with DNS or experimental data for each of the cases.

6.1 Two Dimensional Channel

The $\bar{v}^2 f - k\omega$ model was first tested for predicting the flow through a two-dimensional channel. The results were compared to the DNS data of Moser [3] at $Re_{\tau} = 395$ and $Re_{\tau} = 590$. The computational grid consisted of $141(x) \times 41(y)$ grid points and all $y^+$ values were less than 1. To ensure that the results are independent of the grid, all simulations were repeated with twice the number of grid points in each spatial direction. No differences were observed, therefore the solutions are assumed to be grid independent.

The Figures 6.1-6.3 show computed velocity, kinetic energy, production, and dissipation. The plots are in wall units: thus $y^+ = y Re_{\tau}$, $u^+ = U/u_{\tau}$, $k^+ = k/u^2_{\tau}$, $\varepsilon^+ = \varepsilon/Re_{\tau}$. In Fig. 6.1 all three models accurately predict the correct velocity profile for both $Re_{\tau} = 395$ and $Re_{\tau} = 590$.

In Fig. 6.2 it can be seen that the $k - \omega$ model underpredicts the kinetic energy near the wall. This is typical of the $k - \omega$ model and is due to the fact that $\varepsilon = \beta^* \omega k$ produces an
overprediction of $\varepsilon$ near the wall. Low Reynolds number damping functions can be used to suppress this overprediction of $\varepsilon$, but these damping functions are based on $Re_\tau$ and are typically ill suited for complex flows.

The $\nu^2f - k\omega$ model produces better kinetic energy predictions near the wall because a two-layer approach to computing $\varepsilon$ is used in the $k$ equation. The two-layer approach uses the approximation $\varepsilon = \beta^* \omega k^{0.5} \nu^{2.5}$ if $Re_y > 200$, and uses a mixing length approximation $\varepsilon = k^{3/2}/l_\varepsilon$ if $Re_y < 200$. In these expressions $Re_y = k^{1/2}y/v$ and
\[ l_\varepsilon = C_1 y \left( 1 - e^{-y\sqrt{k/\nu A_\varepsilon}} \right) \] where \( C_1 = \kappa / C_{\mu}^{3/4} \) and \( A_\varepsilon = 2C_1 \). This could be used in the standard \( k - \omega \) model but additional damping functions would be needed in the \( \omega \) equation to increase \( \omega \) near the wall. As pointed out earlier, the term \( \omega^2 \left( \frac{\nu}{k} \right)^{1-n} \) in Eq. 4.33 tends to increase \( \omega \) near the wall which compensates for the increased \( k \) near the wall. In the original \( \omega \) equation, Eq. 4.34, a damping function must be applied to reduce the term \( \omega^2 \) increasing \( \omega \) near the wall. This balance between \( k \) and \( \omega \) is needed so that the eddy viscosity is not overpredicted. Figure 6.3 shows predictions of turbulent production and dissipation. The turbulent production is accurately predicted by each model but as stated above the \( k - \omega \) model and the \( \nabla^2 f - k\omega \) model overpredict the dissipation near the wall. The dissipation predicted by the \( \nabla^2 f - \varepsilon \) model is in good agreement with experiments. Figure 6.4 shows that each version of the \( \nabla^2 f - \varepsilon \) model accurately predicts the law-of-the-wall but overpredicts the kinetic energy at the wall. Version 1 seems to produce the largest overprediction.
6.2 Backward Facing step

The backward-facing step is a common problem used to test turbulence models. The geometry is simple and the separation and reattachment locations have been well documented for several different Reynolds numbers. Two different geometries and Reynolds numbers were solved in the present study, one based on the experiments of Driver [37] and the other based on the experiments of Kasagi [46]. For both cases the simulations were performed with 129 ($x$) x 65 ($y$) grid points. The number of grid points were doubled in each spatial direction and no noticeable difference was observed in the results, therefore the solutions are grid independent. For the Driver [37] case the $k - \varepsilon$ and $k - \omega$ models were first compared for computing the streamwise velocity, kinetic energy, and skin friction. Figure 6.5 shows the model predictions for the mean velocity. It can be seen that the $k - \varepsilon$ model underpredicts the boundary layer recovery downstream from reattachment compared to the $k - \omega$ model. Immediately downstream of the step the flow experiences a large adverse pressure gradient. As pointed out by Wilcox [11], the $k - \varepsilon$
Figure 6.5. Mean velocity predictions for the $k - \varepsilon$ and $k - \omega$ models.

Figure 6.6. Skin friction predictions for the $k - \varepsilon$ and $k - \omega$ models.

model does not correctly predict boundary layer development when subjected to adverse pressure gradients. In his study he tested both models for computing the flow over a flat plate subjected to both adverse and favorable pressure gradients. He found that when the flow experiences an adverse pressure gradient, the $k - \varepsilon$ model overpredicts the skin friction by as much as 40%. This is caused by an underprediction of the boundary layer
development. He also found that the $k - \varepsilon$ model underpredicts the skin friction by about 10% when the flow is subjected to a favorable pressure gradient. He also shows that the $k - \omega$ model accurately predicts the skin friction regardless of whether the flow experiences an adverse or favorable pressure gradient. Fig. 6.6 shows the skin friction predicted by the $k - \varepsilon$ and $k - \omega$ models. The reattachment location is represented by the location where the skin friction is zero. Figure 6.6 shows that the $k - \varepsilon$ model underpredicts the reattachment location whereas the $k - \omega$ model correctly predicts the reattachment location which is at $x/h = 6.4$. Downstream of reattachment, the flow experiences a favorable pressure gradient. As stated above $k - \varepsilon$ models underpredict the skin friction under this condition and is shown in Fig. 6.6. In the bottom corner of the step, corresponding to $x/h = 0$, there is a small secondary eddy formed by the interaction of the large recirculation zone with the wall of the step.

Figure 6.7. Kinetic energy predictions for the $k - \varepsilon$ and $k - \omega$ models
This can be seen in Fig. 6.6 where at $x/h = 0$ the skin friction begins to increase and then decrease at approximately $x/h = 2$. The $k - \omega$ model better represents the eddy size compared to the $k - \varepsilon$ model. Typically $k - \varepsilon$ models overpredict the eddy viscosity in the corner of the step resulting in an underprediction of the eddy size. Figure 6.7 shows the kinetic energy predicted by the $k - \varepsilon$ and $k - \omega$ models. The $k - \omega$ model shows improved predictions upstream of $x/h = 7$ where the $k - \varepsilon$ model underpredicts the kinetic energy. Downstream of $x/h = 7$ the $k - \omega$ model overpredicts the kinetic energy while the $k - \varepsilon$ model is in good agreement with the experiments. This is primarily due to the differences in the diffusion coefficients used in the models.

![Skin Friction (Driver and Seegmiller)](image)

Figure 6.8. Skin friction predictions for the $v^2f - k\varepsilon$ models.

Upon observation of Eq. 4.39 the $k - \omega$ model diffusion coefficient is $\sigma_k = 0.5$ whereas for the $k - \varepsilon$ model $\sigma_k = 1.0$. This causes the kinetic energy predicted by the $k - \varepsilon$ model to diffuse more rapidly as the flow moves downstream instead of maintaining a larger concentration of kinetic energy in the shear layer. Next, all three versions of the $v^2f - k\varepsilon$ model were compared and the mean velocity predictions are shown in Fig. 6.9.
Figure 6.9. Mean velocity predictions for the $v^2 f - k\varepsilon$ models

Figure 6.10. Kinetic energy predictions for the $v^2 f - k\varepsilon$ models

Version 3 of the $v^2 f - k\varepsilon$ model which uses the boundary condition $f_{22w} = 0$ at a solid wall, predicts the mean velocity profiles in good agreement with experiments. Version 1 and 2 underpredict the boundary layer development downstream of reattachment which is
characteristic of the $k - \varepsilon$ model. This is not surprising since the $v^2f - k\varepsilon$ model uses the $k$ and $\varepsilon$ equations.

The kinetic energy predicted by the $v^2f - k\varepsilon$ models is shown in Fig. 6.10. Versions 1 and 3 produce similar kinetic energy predictions upstream of reattachment whereas version 2 produces considerably less kinetic energy. The coefficient $C_{\varepsilon 1}$ in the $\varepsilon$ equation, Eq. 4.29, for version 2 typically takes on values much larger than its counterpart Eq. 4.28 of version 1. This causes the kinetic energy predicted by version 2 in Fig. 6.10 to be lower and decrease more rapidly compared to the other versions. Figure 6.8 shows the skin friction predicted by each version of the $v^2f - k\varepsilon$ models. The skin friction predicted by version 1 shows that the reattachment of the shear layer is accurately predicted, whereas, version 2 overpredicts the reattachment and version 3 underpredicts the reattachment. The overprediction of the reattachment by version 2 is caused because of the large value of $C_{\varepsilon 1}$ mentioned above. This was shown in Fig. 6.10 to produce lower production of kinetic energy which will also result in lower levels of eddy viscosity. This in turn causes the recirculation zone to be overpredicted. Although version 3 shows good agreement with the mean velocity in Fig. 6.8, the reattachment is underpredicted and downstream of reattachment the skin friction is overpredicted. What is interesting to note is that no one version of the $v^2f - k\varepsilon$ model shows to be superior. The skin friction predicted by version 1 is in good agreement with experiments, but version 3 shows the best streamwise velocity predictions.

Next, the $v^2f - k\omega$ model was compared to all three versions of the $v^2f - k\varepsilon$ model
and the $k - \omega$ model, and the mean velocity predictions are shown in Fig. 6.11.

Figure 6.11. Mean velocity predictions for the $v^2f - k\omega$, $v^2f - k\varepsilon$, and $k - \omega$ models.

As seen in Fig. 6.5 for the $k - \omega$ model, the $v^2f - k\omega$ model accurately predicts the boundary layer development. Since the $v^2f - k\omega$ model solves equations for $k$ and $\omega$ it should have similar characteristics as that of the $k - \omega$ model but with improved predictive capabilities. The $v^2f - k\omega$ model has been formulated such that it adopts the desirable characteristics of the $k - \omega$ model but improves predictions where the $k - \omega$ model performs poorly. Similarly, the $v^2f - k\varepsilon$ model has been formulated such that it adopts the desirable characteristics of the $k - \varepsilon$ model but improves predictions where the $k - \varepsilon$ model performs poorly. This can be seen in Fig. 6.8 by observing that the $v^2f - k\varepsilon$ model accurately predicts the reattachment location whereas the $k - \varepsilon$ model underpredicts the reattachment location. Although the $v^2f - k\varepsilon$ improves predictions over the $k - \varepsilon$ model, if the $k - \varepsilon$ model has a severe deficiency then the $v^2f - k\varepsilon$ model may also have some of
the same deficiency. As state earlier the $k - \varepsilon$ model performs poorly in the presence of an adverse pressure gradient. Figure 6.11 shows that the $v^2 f - k\varepsilon$ model also performs poorly under these circumstances. The boundary layer development is underpredicted, which is similar to that of the $k - \varepsilon$ model in Fig. 6.5, but with a slight improvement. The
kinetic energy predicted by the $v^2f - k\varepsilon$, $v^2f - k\omega$, and $k - \omega$ models are shown in Fig. 6.12. The $v^2f - k\omega$ model shows improved predictions at $x/h = 1$, and shows improved results compared to the $v^2f - k\varepsilon$ model for the other locations up to the reattachment location, where all three models produce similar predictions. The skin friction predicted by version 1 of the $v^2f - k\varepsilon$ model, the $v^2f - k\omega$ model, and the $k - \omega$ model is shown in Fig. 6.13. The $v^2f - k\omega$ model accurately predicts the reattachment location and shows improved predictions of skin friction downstream of reattachment. Next, all three versions of the $v^2f - k\varepsilon$ model were compared with the experiments of Kasagi [46]. The Reynolds number based on the upstream centerline velocity and step height is 5541 which is considerably less than the case of Driver [37] which was 35000. In the Driver [37] case

![Streamwise Pressure Gradient](image)

**Figure 6.14.** Streamwise pressure gradient

it was shown that the $k - \varepsilon$ and $v^2f - k\varepsilon$ models poorly predict the boundary layer recovery in the presence of an adverse pressure gradient. In the case of Kasagi [46] the Reynolds number is much less than that of Driver [37] which means that the adverse
pressure gradient is much less. The streamwise pressure gradient for both Kasagi [46] and Driver [37] are shown in Fig. 6.14. Mean velocity predictions are shown in Fig. 6.15 for all versions of the \( v^2f - k\epsilon \) model. The \( v^2f - k\epsilon \) models show good agreement with experiments. Figure 6.14 shows that the streamwise pressure gradient has been reduced by a factor of 1000 compared to that of Driver’s case. Since the pressure gradient is low it seems as though the \( v^2f - k\epsilon \) model predictions are not being affected by the adverse pressure gradient. Version 2 shows a slight underprediction of the boundary layer recovery and is probably due to the fact that \( C_{\epsilon_1} \) in the \( \epsilon \) equation, Eq. 4.29, takes on much larger values than Eq. 4.28 of version 1. This increases the production of \( \epsilon \) causing a reduction in eddy viscosity. Lower eddy viscosity causes the boundary layer to recover slower which was also seen above for Driver [37]. The kinetic energy predicted by the
Figure 6.16. Kinetic energy predictions for the $v^2f - k\varepsilon$ models.

Figure 6.17. Streamwise velocity predictions for the $\overline{v^2}f - k\varepsilon$, and the $\overline{v^2}f - k\omega$ models.
6.3 Cavity

The flow in a grooved turbine blade tip was computed and compared with the experiments of Metzger [47]. The flow is essentially two-dimensional at Reynolds number
\[ \text{Re} = \frac{\rho U_{\text{bulk}} C}{\mu} = 1.5 \times 10^4 \]

where \( U_{\text{bulk}} \) is the bulk fluid entering the cavity and \( C \) is the inlet channel width. The ratio of cavity height, \( d \), to cavity width, \( w \), was chosen to be 0.2.

Figure 6.19. Cavity setup

The incoming flow was at a temperature of approximately 326 \( K \), while the walls were maintained at 316 \( K \). The results presented here were from solutions with 229 \( (x) \times 65 \) \( (y) \) grid points. All \( y^+ \) values were less than 1, and a grid study was performed to ensure that the solutions presented are independent of the grid. Figure 6.20 shows temperature contours for all three versions of the \( \overline{v^2}f - k\varepsilon \) model and the \( \overline{v^2}f - k\omega \) model. Each of the models predict a large circulation region inside of the cavity and a small corner eddy in the lower left corner. The corner eddy predicted by the \( \overline{v^2}f - k\omega \) model is considerably larger than predicted by any of the \( \overline{v^2}f - k\varepsilon \) models. This is similar to the backward facing step predictions where the skin friction of Fig. 6.13 shows that the \( \overline{v^2}f - k\varepsilon \) model underpredicts the corner eddy while the \( \overline{v^2}f - k\omega \) model predictions are in better agreement with the experiments.
The temperature contours predicted by version 1 of the $v^2f - k\varepsilon$ model are more diffuse than the other two versions causing more of the heated inflow to fill the cavity. Figure 6.21 shows the eddy viscosity predicted by all three versions of the $v^2f - k\varepsilon$ model and the $v^2f - k\omega$ model. Version 1 of the $v^2f - k\varepsilon$ model predicts more eddy viscosity in the large recirculation zone compared to the other versions which results in more turbulent diffusion. This causes the hot inflow temperature to diffuse more into the cavity compared with the other versions. Version 1 has typically been shown to produce
Figure 6.21. Eddy viscosity predictions for the $v^2 f - k\varepsilon$ and $v^2 f - k\omega$ models.

better results for most problems compared to the other versions of the $v^2 f - k\varepsilon$ model.

The eddy viscosity predicted by versions 2 and 3 of the $v^2 f - k\varepsilon$ models is similar and is primarily due to the fact that both of these versions use the same expression for computing $C_{\varepsilon1}$ in the $\varepsilon$ equation. The expression for $C_{\varepsilon1}$ for versions 2 and 3 is shown in Eq. 4.29 and was developed so that the wall distance to each grid point is not needed which is used in Eq. 4.28. The expression Eq. 4.29 typically produces more turbulent dissipation, $\varepsilon$, and reduces the eddy viscosity. This was discussed in the previous section for flow
Figure 6.22. \( C_{\varepsilon 1} \) contours

over the backward facing step and was shown to cause version 2 to underpredict the boundary layer development downstream of the step. Figure 6.22 shows computed values of both expressions for \( C_{\varepsilon 1} \). For version 1 \( C_{\varepsilon 1} \) ranges from 1.3 in the outer flow regions, to 1.55 near the wall. The \( C_{\varepsilon 1} \) expression for version 1 uses the distance from each grid point to the nearest wall. Some researchers believe that when coefficient are computed using expressions similar to this, that the predictions are geometry dependent, which is undesirable. In order to eliminate this dependence the expression Eq. 4.29 for version 2 and 3 was developed. Although the wall dependence has been eliminated, version 2 does not typically produce results as accurate as version 1, and is also more numerically stiff than version 1. The version 2 expression of \( C_{\varepsilon 1} \) has been formulated such that it should range between 1.4 in the outer flow region, to about 1.65 near the wall. Figure 6.22 shows that \( C_{\varepsilon 1} \) for version 2 has values of over 2 in the corners and transitions rapidly to lower values along the bottom and side walls. This can add additional instabilities
Figure 6.23. $\bar{v}^2$ predicted by the $\bar{v}^2 f - k\varepsilon$ models and the $\bar{v}^2 f - k\omega$ model to the $\bar{v}^2 f - k\varepsilon$ model. Figure 6.23 shows computed values of $\bar{v}^2$ for each version of the $\bar{v}^2 f - k\varepsilon$ model and the $\bar{v}^2 f - k\omega$ model. Version 1 predicts a larger value of $\bar{v}^2$ compare to the other two versions, but is similar to that of the $\bar{v}^2 f - k\omega$ model.

The scalar equation 4.30 for the velocity $\bar{v}^2$, is formulated such that it represents the normal fluctuating turbulent stress near the wall, but is simply a velocity scale away from walls. The advantage of using $\bar{v}^2$ in the eddy viscosity calculation is that it reduces the overprediction of eddy viscosity predicted by conventional $k - \varepsilon$ models for many
flows. This improvement results because $\overline{v^2}$ is always less than the kinetic energy $k$. The higher values of $\varepsilon$ predicted by version 2 and 3 produces more dissipation for the kinetic energy, $k$, and the scalar velocity $\overline{v^2}$. The $\overline{v^2}f - k\omega$, $k - \varepsilon$, and $\overline{v^2}f - k\varepsilon$ models were compared and the computed Nusselt number predicted by each model is shown in Fig. 6.24. The Nusselt number predicted by the $\overline{v^2}f - k\omega$ model shows the best agreement with experiments. It produces the correct trend accurately and predicts the peak value in good agreement with experiment. The overprediction of the Nusselt number by version 1 of the $\overline{v^2}f - k\varepsilon$ model is caused by an overprediction of the eddy viscosity near the wall. This overprediction of eddy viscosity is caused because the $\overline{v^2}f - k\varepsilon$ models overpredict the kinetic energy near the wall and was shown in Fig. 6.4 for two dimensional flow in a channel. Version 2 also overpredicts the kinetic energy near the wall for a two dimensional channel flow, but the $C_{\varepsilon 1}$ expression for version 2 produces more dissipation near the wall. This reduces the near-wall eddy viscosity and Nusselt number. The $k - \varepsilon$ model predicts the Nusselt number to decrease as the side wall is approached while the experiments show that it increases. After the flow attaches to the bottom surface it experiences a strong adverse pressure gradient caused by the approaching side wall. As pointed out earlier, the $k - \varepsilon$ model performs poorly in the presence of an adverse pressure gradient causing the Nusselt number to decrease too rapidly. All of the models incorrectly predict the Nusselt number below $x/H = 0.5$. Version 3 of the $\overline{v^2}f - k\varepsilon$ model produces the best results compared with the other versions.
6.4 Coaxial Jet

The flow in a coaxial jet was computed using the $k - \omega$, $\overline{v^2} f - k\varepsilon$, and $\overline{v^2} f - k\omega$ models. The flow in the annular part is three times as large as that in the inner pipe. The models are compared with the experimental data of Habib [40]. Figure 6.25 shows the velocity magnitude predictions for all three versions of the $\overline{v^2} f - k\varepsilon$ models, the $\overline{v^2} f - k\omega$ model, and the $k - \omega$ model.
Figure 6.25. Velocity Magnitude
Each version of the $\nu^2 f - k\varepsilon$ model and the $\nu^2 f - k\omega$ model are compared with version 1. Version 2 predicts a slightly longer reattachment location compared to version 1. These predictions are similar to those of the backward facing step where the skin friction in Fig. 6.8 shows that the reattachment location was overpredicted for version 2. Version 3 predicts a shorter reattachment location compared to version 1 which was also seen in the backward facing step flow where Fig. 6.8 showed that version 3 underpredicted the reattachment location. The $\nu^2 f - k\omega$ model predicts a shorter reattachment location compared to version 1 of the $\nu^2 f - k\varepsilon$ model. Figure 6.26 shows centerline velocity predictions. Version 1 of the $\nu^2 f - k\varepsilon$ model accurately predicts the peak velocity along the centerline but slightly overpredicts the centerline velocity downstream of $x/D = 2$. The $\nu^2 f - k\omega$ best represents the velocity decay along the centerline downstream of $x/D = 2$ which would correspond to a more accurate prediction of the reattachment location compared to the other models. Version 2 of the $\nu^2 f - k\varepsilon$ model overpredicts the centerline velocity between $x/D = 2$ and $x/D = 3$, but accurately predicts the centerline velocity downstream of $x/D = 3$. Version 3 of the $\nu^2 f - k\varepsilon$ model underpredicts the peak centerline velocity and overpredicts the spreading rate of the jet resulting in an overprediction of the centerline velocity decay. The $k - \omega$ model underpredicts the spreading rate of the jet which results in an underprediction of the centerline velocity decay. Figure 6.27 shows the eddy viscosity predicted by each of the models. Each model is compared to version 1 of the $\nu^2 f - k\varepsilon$ model because this version was shown by Durbin [48] to produce better results compared to other models.
Upstream of $x/D = 1.8$, version 1 predicts a higher eddy viscosity near the centerline. This causes the annular jet to diffuse faster toward the centerline which is in better agreement with the experiments in Fig. 6.26. Downstream of $x/D = 1.8$, version 2 predicts more eddy viscosity compared with version 1. This improves the centerline velocity prediction in this region by increasing the spreading rate of the jet. Version 3 of the $v^2f - k\varepsilon$ model produces considerably more eddy viscosity compared with any of the models. This causes the spreading rate of the jet to be overpredicted which was also seen for the backward facing step where Fig. 6.8 shows an underprediction of the reattachment.
location. Version 1 of the $v^2 f - k\varepsilon$ model predicts more eddy viscosity near the centerline compared with the $v^2 f - k\omega$ model upstream of $x/D = 2.2$. Downstream of $x/D = 2.2$, the centerline eddy viscosity predicted by the $v^2 f - k\omega$ model is slightly higher. This increases the spreading rate of the jet in this region, and produces better centerline velocity predictions which is seen in Fig. 6.26.

An interesting observation is that for both version 1 and 2, the eddy viscosity becomes wavy at large downstream locations near the wall. This is caused because the turbulent dissipation, $\varepsilon$, and the length scale $L$, Eq. 4.22, have become very small. The small value of $L$ reduces the diffusion in the $f_{22}$ equation causing it to become increasingly source term dominated in which conventional solvers have difficulty solving. Since $\varepsilon$ has also become small, the wall boundary condition $f_{22w}$, Eq. 4.27, becomes large causing increasing instability in the $f_{22}$ equation. Version 3 does not show this behavior but excessively overpredicts the eddy viscosity. This causes the flow to rapidly reattach to the wall which was shown in Fig. 6.26 and is not in agreement with experiments. The eddy viscosity predicted by the $v^2 f - k\omega$ model does not show the wavy eddy viscosity near the wall like the $v^2 f - k\varepsilon$ model and produces reasonable levels of eddy viscosity to correctly predict the reattachment location and decay of the centerline velocity. The streamwise velocity predictions at various locations for each model are shown in Fig. 6.28. Version 1 of the $v^2 f - k\varepsilon$ model and the $v^2 f - k\omega$ model produce similar results at $x/D = 0.616, 1.43$, and $2.23$ but the $v^2 f - k\omega$ shows better agreement with the experiments at $x/D = 3.67$. 
Figure 6.27. Eddy viscosity predictions
Although the experiments do not show the exact location of the reattachment location, the fact that the $v^2f - k\omega$ model shows improved predictions at $x/D = 3.67$ indicates that the $v^2f - k\omega$ model better represents the reattachment location and boundary layer development. The other versions of the $v^2f - k\epsilon$ model show good agreement with the experiments at $x/D = 0.616$, but show that the spreading rate of the jet is underpredicted at $x/D = 1.43$. For version 3, it was seen in Fig. 6.27 that the eddy viscosity is overpredicted causing the jet to rapidly reattach to the wall producing a velocity profile at $x/D = 3.67$ similar to that of a fully developed pipe flow.

Figure 6.28. Streamwise velocity predictions
Figure 6.29. Kinetic energy predictions
The kinetic energy predicted by each of the models is shown in Fig. 6.29. Once again, each model is compared to version 1 of the $\overline{u'^2}f - k\varepsilon$ model. The kinetic energy predicted by version 2 of the $\overline{u'^2}f - k\varepsilon$ model is slightly less than that of version 1 although Fig. 6.27 shows that the eddy viscosity is higher for version 2. This means that Eq. 4.29 for $C_{\varepsilon 1}$ produces a greater decay of turbulent dissipation as the flow moves downstream compared to Eq. 4.28 for version 1. The kinetic energy predicted by version 3 shows a rapid decay after $x/D = 3$ compared to that of version 1. Since the velocity predicted by version 3 reattaches to the wall rapidly, the jet spreads much faster, causing the turbulent production produced by the large velocity gradients in the jet to decay and reduce the kinetic energy. The kinetic energy predicted by the $\overline{u'^2}f - k\omega$ model shows slightly larger values compared to version 1 which produces the increase in eddy viscosity seen in Fig. 6.27. This was shown to produce better centerline velocity predictions by the $\overline{u'^2}f - k\omega$ model in Fig. 6.26. Figure 6.30 shows the centerline kinetic energy predicted by each model. All versions of the $\overline{u'^2}f - k\varepsilon$ model and the $\overline{u'^2}f - k\omega$ model overpredict the kinetic energy upstream of $x/D = 1.2$. Downstream of this the $\overline{u'^2}f - k\omega$ model shows better agreement with the experiments. Figure 6.31 shows computed kinetic energy profiles at various downstream locations for each model. At $x/D = 0.616$ version 1 of the $\overline{u'^2}f - k\varepsilon$ model shows good agreement with experiments except near the wall. In the near wall region both versions of the $\overline{u'^2}f - k\varepsilon$ model overpredict the kinetic energy near the wall. This was also seen for the fully developed channel flow of Fig. 6.4 where the $\overline{u'^2}f - k\varepsilon$ model overpredicts the kinetic energy near the wall.
At $x/D = 1.43$, all of the models overpredict the kinetic energy in the bulk of the flow but produce reasonable results near the centerline. Once again both versions of the $v^2f - k\epsilon$ model overpredict the kinetic energy near the wall. At $x/D = 2.23$, the main difference between the models occurs near the centerline where the $v^2f - k\omega$ model and version 1 of the $v^2f - k\epsilon$ model produce the best results.
Figure 6.31. Kinetic energy predictions
Chapter 7. Jet-in-Crossflow

External film cooling of turbine blades involves the injection of a coolant jet over the blade surfaces or the end walls (hub and blade tip). These flows are difficult to predict accurately due to the inherent complexity of the jet-crossflow interaction. Figure 7.1 shows a cartoon from Fric [49] illustrating the various structures generated when a jet is injected normally into an unbounded crossflow. Unlike a rigid cylinder in crossflow, the boundaries of the jet are compliant and entraining, causing the jet to bend over. Periodic shedding of wake vortices have been observed particularly when the jet blowing ratio \( V_{\text{jet}}/V_\infty \) is greater than Fric [49]. Considerable effort has gone into determining the origin of the wake vortices, and there is now experiments from Fric [49], Kelso [50], and Andreopoulos [51] and computational evidence from Tyagi [52] that the wake vortices are initiated by the entrainment of the crossflow boundary layer into the wake, and the upward reorientation of the entrained flow into the wake structures. The jet structure itself is dominated by a pair of kidney shaped counter-rotating vortex pair (CVP), and both the shearing between the jet and the crossflow and the vorticity issuing from the jet exit has been attributed to be the source of the CVP, Andreopoulos [51]. There are however different mechanisms proposed on the reorientation of the jet-hole vorticity into the CVP structure. Kelso [50] and Andreopoulos [51] show that upstream of the jet, due to the adverse pressure gradients, a horse-shoe vortex system is formed, which wraps around the base of the jet travelling downstream with vorticity counter to the CVP. Shear layer vortices on the leeward and windward edges of the jet have also been observed by Kelso.
[50], and have been attributed to Kelvin-Helmholtz instabilities. These and other studies provide unambiguous evidence of the importance of the coherent structures and their dynamics in the near field of the injected jet. Clearly, any predictive model must embody the physics of the coherent structures to accurately predict the near-field jet behavior.

This chapter focuses attention on the jet-in crossflow configuration representative of film cooling. The main goal of this chapter is to demonstrate how the new $\overline{v^2f - k\omega}$ model predicts this kind of flow and to compare the results with some existing model as well as experimental data.

![Figure 7.1. Schematic of the flow field of a jet in a crossflow](image)

### 7.1 Previous Studies

The majority of the RANS simulations for jet in a cross flow have employed a variant of the $k - \epsilon$ models (originally proposed by Launder and Spalding [53]) to obtain the distribution of eddy viscosity. Patankar et al.[54] were among the early researchers to use this model to perform a detailed study of the jet in a cross flow, and even with a relatively
coarse (15x15x10) grid, obtained reasonable agreement with experimental data for the jet trajectory and streamwise velocity. Jones and McGuirk [55] used a grid containing 20x15x15 nodes and obtained only qualitative agreement with measured data due to the inadequate grid resolution. Grid resolution requirements were investigated by Demuren [56] in his computations for a row of jets in a crossflow. Results on a 37x70x14 (streamwise, vertical and spanwise directions) were shown to be grid independent and captured experimental trends fairly well. Demuren [57] also published a detailed analysis on modeling turbulent jets in cross flow, and presented a systematic review of the various models reported till 1985.

Claus and Vanka [58] used a refined grid (256x96x96) and the $k-\varepsilon$ model and found that they could not capture the horseshoe vortex. Kim and Benson [59] employed a multiple-time-scale turbulence model to perform a detailed analysis of the flowfield of a row of jets in a confined cross flow. The horseshoe structure was predicted correctly using a non-uniform 165x59x80 grid and the good agreement was attributed partly to the multiple-time-scale model used for this study. An analysis of cooling jets near the leading edge of turbine blades was performed by Benz et al.[60]. The RANS equations coupled with the standard $k-\varepsilon$ model was solved. A multi-block grid was used to simulate an actual blade geometry along with the coolant supply hole. Good agreement with experimental results was obtained due to the inclusion of the coolant delivery tube along with the main flow.

Garg and coworkers have systematically studied the effects of turbulence models [61]
and the hole physics [62],[63],[64],[65]. In Garg [61], an ACE rotor with five rows containing 93 film cooling holes were simulated. Three different turbulence models were explored (Wilcox’s $k - \omega$, Coakley’s $q - \omega$ and the Baldwin-Lomax model). Results were compared with the experimental data of Abhari [66]. Overall, the $k - \omega$ model appears to provide the best agreement with the measurements, and particularly on the pressure side. Garg and Rigby [62] used Wilcox’s $k - \omega$ turbulence model, and found that the coolant velocity and temperature profiles at the hole exit did not conform to the commonly used parabolic or $1/7$ power law distribution. The exit velocity profile appeared to significantly impact the heat transfer coefficient distribution on the suction side, and to obtain reasonable predictions, it was shown that the flow development in the coolant delivery tube must be accounted for. In another application of Wilcox’s $k - \omega$ turbulence model, Garg [63] computed heat transfer coefficient on the blade, hub and shroud for a rotating high-pressure turbine blade with 172 film-cooling holes in eight rows.

Leylek and Zerkel [67] included the coolant supply hole and the plenum in their calculations and used the standard $k - \varepsilon$ model employing generalized wall functions prescribed by Launder and Spalding [53]. York and Leylek [68] presented predictions for mainstream pressure gradient effects in film cooling. A realizable $k - \varepsilon$ model was used and the computations demonstrated the ability of the applied computational methodology to accurately model film cooling in the presence of mainstream pressure gradients. A detailed analysis of film-cooling physics, in a four part series, has been presented by Walters and Leylek [69], McGovern and Leylek [70], Hyams and Leylek [71], and
Brittingham and Leylek [72], each dealing with different aspects of the film cooling problem. The standard $k - \varepsilon$ model employing wall functions and a two layer model was used.

An aerodynamic and heat transfer analysis of discrete site film-cooled turbine airfoils was conducted by Edwards et al. [27] (1994). Ajersch et al. [73] made detailed measurements of multiple square jets injected normally into a cross flow and carried out an accompanying numerical simulation using a multi-grid, segmented, $k - \varepsilon$ CFD code. Predictions and measurements did not compare well for velocities and stresses on the jet centerline, while values off the centerline matched those of the experiments much more closely. A similar study for an inclined jet was performed by Findlay et al. [74]. A numerical study of discrete-hole film cooling was conducted by Berhe and Patankar [75] on a three dimensional film cooling geometry that included the main flow, injection hole and the plenum. The effect of various variables like blowing ratio, density ratio, hole length, plenum height, plenum flow direction and turbulence level at inlet were discussed in detail. Berhe and Patankar [76] extended their flat plate studies and included the effect of curvature using a Richardson type correction and a two-equation model. The standard $k - \varepsilon$ and the two-layer $k - \varepsilon$ turbulence models were used by Lakehal et al. [77] for investigating film cooling effectiveness of a flat plate by a row of laterally injected jets. In order to match the measured lateral spreading, they employed an anisotropic correction for eddy viscosity proposed by Bergeles et al. [78]. Hoda and Acharya [79] compared seven different turbulence models for film cooling flows and concluded that the Lam-Bremhorst
formulation provided the best comparison with the measurements. More complex configurations have also been studied. A transonic film cooling investigation of the effect of hole shapes and orientations was carried out by Wittig et al.[81]. Bohn et al.[82] made detailed 3-D conjugate flow and heat transfer calculations of a film-cooled turbine guide vane at different operational conditions. More recently, Heidmann et al.[83] have reported a fully coupled calculations of an Allied-Signal film cooled vane with shaped holes. Their calculations included both the internal cooling channels, the coolant delivery tubes, and the external flow. In these cases, the model predictions were only in qualitative agreement with data, and a need for improved turbulence modeling is clearly required.

Figure 7.2. Computational grid (left) and domain decomposition (right)

The film cooling configuration and parameters selected correspond to the measurements of Lavrich and Chiapetta [84]. Results with the $\overline{v^2}f - k\omega$ model are compared to the high Reynolds number $k - \omega$ model, the two-layer $k - \varepsilon$ model, version 1 of the $\overline{v^2}f - k\varepsilon$
model, and experimental measurements. In these simulations, the blowing ratio \( \left( V_{\text{jet}} / V_{\infty} \right) \) is one. The jet inlet temperature is 310.78 K and the crossflow temperature is 288 K. The Reynolds number based on the jet velocity and jet diameter is 22200. The velocity and temperature measurements were taken using a hot wire, while the adiabatic effectiveness was measured using heat sensitive paint and an infrared camera. The computational domain extended 5D upstream of the hole, 13D downstream of the hole, and 3.5D in the spanwise direction. The height of the domain extended to 5D above the hole. The computational grid is shown in Fig. 7.2 and consists of 1.3 million grid points. The grid was created using GridPro and each colored block on the left represents a different zone. The grid in Fig. 7.2 only shows the grid zones which outline the solid surfaces. The simulations were also computed using 0.7 million grid points. No noticeable differences in the solution were observed, hence the solutions presented here are assumed to be grid independent. The results presented in this section were performed using the 1.3 million grid point domain.

Each simulation was carried out in parallel across 15 PC’s with 1.1 Ghz. Athalon processors. The domain decomposition can be seen in Fig. 7.2 on the right hand side and shows the outline of each zone in the domain. The zones have been colored such that zones of the same color reside on the same processor. Since the domain is symmetric about the jet centerline, \( z = 0 \), only one half of the domain in Fig. 7.2 was simulated. The other half is shown only to represent the complete configuration being simulated.

Figure 7.3 shows contours of the \( \bar{v}^2 f - k \omega \) model predictions of streamwise velocity
compared with the experiments. The left hand side, $z < 0$, represent the computations and the right hand side, $z > 0$, represents the experiments. The experiments do not extend down to the surface but instead, start at $D/10$ above the surface. This is why the boundary layer development seen on the computational side is not seen on the experimental side.

The predictions show, like the experiments, that the jet exhausts into the crossflow and forms a plume that spreads as it moves downstream. It also shows that the model compares quite well with the experiments in that the inner core of the jet and the boundary layer development seem to be accurately predicted.

Figure 7.3. Streamwise velocity prediction
Figure 7.4. Streamwise velocity at x/D = 5
Figure 7.4 shows a closer view of the streamwise contours at $x/D = 5$ for each of the model predictions. The $k - \omega$ model predictions in Fig. 7.4 show that the core of the jet is located at a higher position off the surface compared with the experiments. The $k - \varepsilon$ model predictions show the core of the jet to be at a position slightly lower than experiments. The predictions of the $v^2f - k\varepsilon$ and $v^2f - k\omega$ models are in good agreement with the experiments with very little mis-match between the contours. A careful examination of the contours clearly indicates that the $v^2f - k\omega$ model exhibits better agreement with the data than any of the other models.

Figure 7.5. Streamwise velocity predictions at $z/D = 0$
Figure 7.5 shows the streamwise velocity predictions of each model at the individual locations labeled a-c in Fig. 7.3. The locations a, b, and c correspond to the spanwise centerplane \((z/D = 0)\) and streamwise locations of \(x/D = 0, 5\) and 10. Each of the locations shown in Fig.7.5 has been scaled for clarity. The scaling was done by adding a constant value of 10 to the velocity at each successive \(x/D\) location. Therefore, the scale along the abscissa does not reflect the actual value of the velocity.

At \(z/D = 0\), the \(k - \omega\) model overpredicts the jet-penetration (relative to the other models) at both \(x/D = 5\) and \(x/D = 10\). This was also observed in Fig. 7.4. The other
model predictions are relatively close to each other with the $v^2f - k\omega$ model showing the best agreement with experiments at $x/D = 5$.

Figure 7.6 shows the streamwise velocity predictions of each model at the individual locations labeled d-f in Fig. 7.3. The locations d-f correspond to a spanwise plane at the edge of the jet hole ($z/D = 0.44$) and streamwise locations of $x/D = 0, 5$ and 10. The velocities have been scaled as stated above for Fig. 7.5. All of the models produce similar and satisfactory predictions at this spanwise location of the flow.

Figure 7.7 shows the streamwise velocity predictions of each model at the individual locations labeled d-f in Fig. 7.3. The locations d-f correspond to a spanwise plane at the edge of the jet hole ($z/D = 0.44$) and streamwise locations of $x/D = 0, 5$ and 10. The velocities have been scaled as stated above for Fig. 7.5. All of the models produce similar and satisfactory predictions at this spanwise location of the flow.

Figure 7.7. Streamwise velocity predictions at $z/D = 0.88$
locations labeled g-i in Fig. 7.3. The locations g-i correspond to a spanwise plane away from the jet hole \((z/D = 0.88)\) and streamwise locations of \(x/D = 0, 5\) and 10. This spanwise plane is far away from the hole such that there is no influence from the jet and the velocity profiles are representative of a turbulent flat plate flow. All of the models are in good agreement with the experiments.

Figure 7.8 shows contours of the model predictions of the vertical velocity compared with the experiments. The left hand side, \(z < 0\), represent the computations and the right hand side, \(z > 0\), represents the experiments. As stated for the streamwise velocity, the

![Vertical Velocity for Blowing Ratio = 1](image_url)

Figure 7.8. Vertical velocity prediction
Figure 7.9. Vertical velocity predictions at x/D = 5
experiments do not extend down to the surface but instead, start at $D/10$ above the surface. Figure 7.8 shows that the $\vec{v}^2f - k\omega$ model predictions are qualitatively in good agreement with the experimental data.

Figure 7.9 shows the details of the vertical velocity predictions from all of the models at $x/D = 5$. The $k - \omega$ model predictions show that the vertical velocity is appreciably overpredicted. This explains why the core of the jet seen in Fig. 7.4 is predicted at a higher location off the surface than in the experiments. The $k - \varepsilon$ model also overpredicts the vertical velocity, but the magnitude of the overprediction is smaller, and a careful comparison of the mis-match between the contours indicate that the $k - \varepsilon$ predictions are in closer agreement with the measurements relative to the $k - \omega$ model predictions. Both the $\vec{v}^2f - k\varepsilon$ and the $\vec{v}^2f - k\omega$ models predict lower vertical velocities, and the model predictions are in much better agreement with the data. The counter rotating vortex pair is characterized by the circular contours on the right (measured) and left (predicted) hand sides of the centerline. The $k - \varepsilon$ model shows more concentrated contours in this region with a greater spanwise extent than that of the $k - \omega$ model indicating a larger overprediction of the circulation rate or the strength of the CVP. The $\vec{v}^2f - k\varepsilon$ model shows better predictions of vertical velocity along the centerline compared to the $k - \omega$ and $k - \varepsilon$ models but also shows an overprediction of the circulation rate of the counter rotating vortex pair. The $\vec{v}^2f - k\omega$ model produces the best results since the vertical velocity along the centerline and the circulation rate of the counter rotating pair are in better agreement with the experiments. Figure 7.10 shows the vertical velocity predictions
of each model at \( z/D = 0 \) at the individual locations labeled a-c in Fig. 7.8. As was shown in Fig. 7.9, the \( k - \omega \) model produces the largest overprediction of vertical velocity compared to the other models. The \( k - \varepsilon \) model also overpredicts the peak vertical velocity at all \( x/D \) locations. As in Fig. 7.9, the best agreement with the data comes from the \( \overline{v^2f} - k\omega \) model. Figure 7.11 shows the vertical velocity at \( z/D = 0.44 \), at the individual locations labeled d-f in Fig. 7.8. All of the models agree well with the experimental data except the \( k - \omega \) model which exhibits significant differences with the measured velocity at \( x/D = 5 \). Figure 7.9 shows the vertical velocity predictions at the locations
Figure 7.11. Vertical velocity predictions at $z/D = 0.44$

labeled g-i in Fig. 7.8. At $x/D = 5$ the $k - \varepsilon$ model overpredicts the peak vertical velocity significantly. The $\overline{v^2} f - k\varepsilon$ model shows improvements over the $k - \varepsilon$ model but still shows a slight overprediction at $x/D = 5$. The $\overline{v^2} f - k\omega$ model shows the best agreement with experiments at all of the locations compared with the other models. Figure 7.3 shows the temperature contours of the $\overline{v^2} f - k\omega$ model compared with experiments. At $x/D = 5$ the model overpredicts the temperature in the core of the jet. However, there is general agreement in the extent of the vertical and lateral spreading of the jet.

Figure 7.14 shows the temperature contours for all the model predictions at $x/D = 5$. 
The $k - \omega$ model predictions show that the temperature is significantly overpredicted in the core of the jet. The other models also show an overprediction of temperature in the core region but the extent of overprediction is considerably lower, particularly along the centerline. All of these models utilized a constant Prandtl number of 0.9. This is the typical value used in turbulence models to compute the temperature field. The value of 0.9 was found experimentally to correspond to the logarithmic region of the boundary layer for flow in a channel. It should not however be considered to be a universal value since the turbulent Prandtl number is a property of the flow. Since Figs. 7.3-7.12 show
that each velocity component predicted by the $v^2f - k\omega$ model is in good agreement with experiments, and Fig. 7.13 and Fig. 7.14 show that there are relatively large discrepancies between the temperature predictions and experiments, this indicates that the constant Prandtl number assumption may not be appropriate in such a flow. In the rest of the paper, attention is focused on developing a suitable expression for the turbulent Prandtl number.

He et al. [85] studied the effect of turbulent Schmidt number on scalar mixing in a jet-in-crossflow. In their study the governing equation for species concentration was identical to the equation for enthalpy, therefore their turbulent Schmidt number should be

![Temperature contours predicted by the $v^2f - k\omega$ model](image)

Figure 7.13. Temperature contours predicted by the $v^2f - k\omega$.
Figure 7.14. Temperature contours at x/D = 5
considered to be the same as the turbulent Prandtl number. They compared a number of simulations for turbulent Schmidt number ranging from 0.2 to 1.5 and a jet-to-crossflow velocity ratio of 2.3. They concluded that the turbulent Schmidt number in the jet should be 0.2 for best results. This conclusion was made because the decay of the thermal plume as the jet moves downstream is in good agreement with experiments for a turbulent Schmidt number of 0.2. A semi-empirical analysis was carried out to suggest a variable turbulent Schmidt number throughout the flow field. Kamotani and Greber [86] developed correlations for the turbulent Schmidt number for a jet-in-crossflow and show that the turbulent Schmidt number increases with increasing momentum flux ratio and density ratio, and with increasing $x/D$. This suggests that the turbulent Schmidt number, or turbulent Prandtl number, should be a variable instead of a constant. As shown by Moffatt and Kays [87] the turbulent Prandtl number in a channel flow takes on a value of about two near the wall and decreases to about 0.85 in the center of the channel. Dunn [88] found that for jet flows the turbulent Prandtl number can take on values as low as 0.2. Chamber [89] found that the turbulent Prandtl number for a turbulent plane jet is about 0.4 in regions of the flow where the Reynolds stresses and heat fluxes are large. Since the jet in a crossflow is a combination of both a jet flow and a wall bounded flow the turbulent Prandtl number should vary between about two or three near the wall and decrease to about 0.2 in the jet region. The following Prandtl number formulation of Moffatt and Kays [87] was initially used in the present study. It was formulated to match the turbulent Prandtl number variation in a channel flow, which ranges between 1.7 near the wall, to
0.85 in the fully turbulent region of the boundary layer.

\[
Pr_t = \frac{1.7}{1 + 0.4Pe_t + 0.08 (\exp(-5/Pe_t) - 1) Pe_t^2}
\]  
(7.1)

No noticeable difference in the solution shown in Figs. 7.14 was observed when this expression was used. This observation indicated that the lower limit of 0.85 in the jet region was inappropriate and that the value should be much lower as pointed out by He [85] and Kamotani and Gerber [86].

In view of the need to define a \(Pr_t\) expression that spanned the appropriate range of values observed experimentally, the following expression for the turbulent Prandtl number was developed in this study for the jet in a crossflow.

\[
Pr_t = \frac{10}{1 + 0.08Re_y}; Re_y = \frac{\rho \sqrt{k_y}}{\mu}
\]  
(7.2)

Figure 7.15 shows contour plots for the turbulent Prandtl number computed by Eq. 7.2 at \(x/D = 5\) and 10. In the core of the jet the turbulent Prandtl number is approximately 0.2
Figure 7.16. Turbulent Prandtl number compared with the suggested value of He et al.

Figure 7.17 shows the temperature contours predicted by the $\overline{v^2 f - k\omega}$ model at $x/D = 5$ for both a constant Prandtl number and with the variable Prandtl number calculation of Eq. 7.2. The contours show that the spreading rate for the predictions with the variable Prandtl number are in much better agreement with experiments compared to the predictions with a constant Prandtl number. Although the temperature is still overpredicted in the core of the jet the size of the overpredicted region has been reduced compared to the constant Prandtl number simulations. This indicates that the variable
Prandtl number expression proposed here moves the predictions in the right direction, but further model developments are needed.

Figure 7.18 shows the temperature predictions for each model at the individual locations labeled a-b in Fig. 7.13. Results are shown at two streamwise locations, $x/D = 5, 10,$ and for a spanwise station, $z/D = 0$. When a constant value of 0.9 is used all of the models overpredict the temperature in the core of the jet, but the predictions with the variable Prandtl number (Fig. 7.18) show better agreement with the experiments.

Figure 7.19 shows the temperature predictions for each model at the individual locations labeled c-d in Fig. 7.13. Results are shown at two streamwise locations, $x/D = 5, 10,$
Figure 7.18. Temperature predictions by each model at $z/D = 0$

Figure 7.19. Temperature predictions by each model at $z/D = 0.44$
Figure 7.20. Temperature predictions by each model at $z/D = 0.88$ and for a spanwise station, $z/D = 0.44$. This spanwise location corresponds to the edge of the jet. At both locations all of the models overpredict the temperature but the $\bar{v}^2 f - k\omega$ model does show some improvement. Despite this improvement, further model developments are needed. The overprediction indicates that the temperature spreading rate in the core of jet is underpredicted.

Figure 7.17 shows the temperature predictions for each model at the individual locations labeled e-f in Fig. 7.13. At these locations the $\bar{v}^2 f - k\omega$ model with variable Prandtl number shows better agreement with experiments. The other models underpredict the temperature indicating that the jet spreading rate is underpredicted. Figure 7.22
Figure 7.21. Adiabatic effectiveness for each model; $k - \epsilon$ (first), $k - \omega$ (second), $\bar{v}^2 f - k\epsilon$ (third), $\bar{v}^2 f - k\omega$ (fourth; Prt=0.9), $\bar{v}^2 f - k\omega$ (fifth; Prt from Eq. 7.2)

shows the centerline adiabatic effectiveness. All of the models overpredict the adiabatic effectiveness when a constant Prandtl number is used while the $\bar{v}^2 f - k\omega$ model with a variable Prandtl number shows good agreement with experiments. When using Eq. (7.2) the Prandtl number takes on values of three to four near the wall reducing the temperature diffusion in this region. This considerably improves the predictions.

Figure 7.23 shows the spanwise variation of adiabatic effectiveness computed by each of the turbulence models using a constant turbulent Prandtl number of 0.9. All of the models overpredict the adiabatic effectiveness when using a constant turbulent Prandtl number. When Eq. (7.2) is used the adiabatic effectiveness is in good agreement with experiments. This clearly points to the importance of using a variable Prandtl number in film cooling computations.
Centerline Adiabatic Effectiveness
Blowing Ratio = 1.0

Figure 7.22. Centerline adiabatic effectiveness

Figure 7.23. Spanwise variation of adiabatic effectiveness
Chapter 8. Conclusions

In the present work six different two equation models were tested for predicting the flow in a stirred tank reactor. The mean velocity fields computed using the six models are compared with experimental LDV data. This was the first study on STRs which examined the performance of DNS-based low-Re $k - \varepsilon$ models. Some specific observation made were: 1) The radial velocity component in the impeller discharge region is overpredicted by each of the models. 2) The tangential velocity component in the impeller discharge region is predicted well by the models, but is underpredicted near the shaft. 3) The LKE model is the only model which produces reasonable kinetic energy predictions in the impeller discharge region.

Each model captures the qualitative circulation patterns in the STR. However, all of the models overpredict the mean radial discharge of the impeller due to an underprediction of the eddy viscosity. The experiments of Dong [22], show that the flow in this region is non-isotropic. To account for the anisotropy in the flow, more sophisticated turbulence models must be employed. The recent work at CTR Verzicco [35] suggest that more than one blade segment on the impeller must be modeled to capture the effects of neighboring blade vortex interactions.

Based on this work a new model, the $\nu^2f - k \omega$ model, was developed and shown to improve the predictions for several different flows compared with conventional $k - \varepsilon$ models, without the need for ad hoc. "damping functions". It also has shown to be numerically robust and does not suffer from the numerical stability problems of the
\( \bar{v}^2 f - k\varepsilon \) model. The new model was first tested for predicting flow in a channel. The results show that the \( \bar{v}^2 f - k\omega \) model improves the near wall turbulence predictions compared to other models and accurately predicts the law-of-the-wall. Next, the \( \bar{v}^2 f - k\omega \) model was tested for computing the flow over two different backward facing step configurations. The predictions were compared with the experiments of Driver [37] and those of Kasagi [46]. The \( \bar{v}^2 f - k\omega \) model was also compared to the \( \bar{v}^2 f - k\varepsilon \) model of Durbin [36] and the \( k - \omega \) model of Wilcox [11]. For the Driver [37] case the adverse pressure gradient downstream of the step causes the \( \bar{v}^2 f - k\varepsilon \) model to underpredict the boundary layer development. This is characteristic of \( k - \varepsilon \) type models and shows that the \( \bar{v}^2 f - k\varepsilon \) model suffers from some of the same deficiencies as the \( k - \varepsilon \) model. The \( \bar{v}^2 f - k\omega \) model performs quite well for both backward facing step configurations. The kinetic energy, streamwise velocity, skin friction and reattachment location are accurately predicted. There are several different versions of the \( \bar{v}^2 f - k\varepsilon \) model. The different versions have been developed in an effort to improve numerical convergence of the \( \bar{v}^2 f - k\varepsilon \) model. All of these versions were compared for both the channel flow and the backward facing step, and no one version has shown to be superior to the others.

Next, the \( \bar{v}^2 f - k\omega \) model was used to compute the flow in a heated cavity. The predictions were compared to the experiments of Metzger [47]. Also, a comparison is made with the different versions of the \( \bar{v}^2 f - k\varepsilon \) model and with the \( k - \varepsilon \) and \( k - \omega \) models. The results show that the \( \bar{v}^2 f - k\omega \) model produces better results compared to the other models in that it predicts the correct trend of the Nusselt number along the bottom
surface of the cavity, and the peak value is accurately predicted near the downstream wall.

The flow in a confined coaxial jet was computed using the $\overline{v^2}f - k\omega$ model and the results were compared with the experimental data of Habib [40]. Also, a comparison was made with the different versions of the $\overline{v^2}f - k\varepsilon$ model and the $k - \omega$ model. Once again the $\overline{v^2}f - k\omega$ model shows good agreement with experiments for predicting the correct decay rate of the jet.

Lastly, the $\overline{v^2}f - k\omega$ model was used to compute the flow in a jet in crossflow. The results were compared with the experiments of UTRC and with the $k - \varepsilon$, and $k - \omega$ models and version 1 of the $\overline{v^2}f - k\varepsilon$ model. The results show that the $\overline{v^2}f - k\omega$ model performs the best for predicting the wake region of the jet. The $k - \omega$ model performs the worst because it grossly overpredicts the eddy viscosity on the jet. All of the models overpredict the adiabatic effectiveness when a constant turbulent Prandtl number of 0.9 is used. This is the typical value used by researchers when using RANS models. Moffatt [87] showed that the turbulent Prandtl number ranges from about two near the wall of a channel flow to approximately 0.9 in the fully turbulent part of the boundary layer. It has also been shown that the turbulent Prandtl number can take on values as low as 0.1 for turbulent jet flows. Based on this a new expression was developed for computing a variable turbulent Prandtl number which ranges from four near the wall, to about 0.2 in the core of the jet. When this expression is used along with the $\overline{v^2}f - k\omega$ model the predictions of adiabatic effectiveness are in good agreement with the experiments.
References


Vita

Raymond Michael Jones was born on November 11, 1971, in Mount Vernon, Texas. After completing his high school education in 1990, he moved to Baton Rouge, Louisiana, to attend college at Louisiana State University. In 1996 he received the degree of Bachelor of Science in Mechanical Engineering. In 1996 he began his graduate studies under the direction of Dr. Sumanta Acharya. During his doctoral program he was funded by the Dow Chemical Company and a grant by LEQS. He received the degree of Doctor of Philosophy in mechanical engineering in August of 2003.