ASSESSING THE ν^2 - f TURBULENCE MODELS FOR CIRCULATION CONTROL
APPLICATIONS

A Thesis

Presented to

the Faculty of California Polytechnic State University,

San Luis Obispo

In Partial Fulfillment

of the Requirements for the Degree of

Master of Science in Aerospace Engineering

by

Travis Marshall Storm

April 2010

© 2010

Travis Marshall Storm

ALL RIGHTS RESERVED

COMMITTEE MEMBERSHIP

TITLE: Assessing the v^2 - f turbulence models for circulation control applications.

AUTHOR: Travis Marshall Storm

DATE SUBMITTED: April 2010

COMMITTEE CHAIR: Dr. David Marshall

COMMITTEE MEMBER: Dr. Robert McDonald

COMMITTEE MEMBER: Dr. Russell Westphal

COMMITTEE MEMBER: Dr. Kim Shollenberger

ABSTRACT

Assessing the ν^2 - f Turbulence Models for Circulation Control Applications

Travis Marshall Storm

In recent years, airports have experienced increasing airport congestion, partially due to the hub-and-spoke model on which airline operations are based. Current airline operations utilize large airports, focusing traffic to a small number of airports. One way to relieve such congestion is to transition to a more accessible and efficient point-to-point operation, which utilizes a large web of smaller airports. This expansion to regional airports propagates the need for next-generation low-noise aircraft with short take-off and landing capabilities. NASA has attacked this problem with a high-lift, low-noise concept dubbed the Cruise Efficient Short Take-Off and Landing (CESTOL) aircraft. The goal of the CESTOL project is to produce aircraft designs that can further expand the air travel industry to currently untapped regional airports.

One method of obtaining a large lifting capability with low noise production is to utilize circulation control (CC) technology. CC is an active flow control approach that makes use of the Coanda effect. A high speed jet of air is blown over a wing flap and/or the leading edge of the wing, which entrains the freestream flow and effectively increases circulation around the wing.

A promising tool for predicting CESTOL aircraft performance is computational fluid dynamics (CFD,) due to the relatively low cost and easy implementation in the design process. However, the unique flows that CC introduces are not well understood, and traditional turbulence modeling does not correctly resolve these complex flows (including

high speed jet flow, complex shear flows and mixing phenomena, streamline curvature, and other challenging flow phenomena). The recent derivation of the ν^2 - f turbulence model shows theoretical promise in increasing the accuracy of CFD predictions for CC flows, but this has not yet been assessed in great detail. This paper presents a methodical verification of several variations on the ν^2 - f turbulence model. These models are verified using simple, well-understood flows. Results for CC flows are compared to those obtained with more traditional turbulence modeling techniques (including the Spalart-Allmaras, k - ϵ , and k - ω turbulence models). Wherever possible, computed results are compared to experimental data and more accurate numerical methods.

Results indicate that the ν^2 - f turbulence models predict some aspects of circulation control flow fields quite well, in particular the lift coefficient. The linear ν^2 - f , nonlinear ν^2 - f , and nonlinear ν^2 - f -cc turbulence models have generated lift coefficients within 19%, 14%, and -26%, respectively of experimental values, whereas the Spalart-Allmaras, k - ϵ , and k - ω turbulence models produce errors as high as 85%, 36%, and 39%, respectively. The predicted stagnation points and pressure coefficient distributions match experimental data roughly as well as standard turbulence models do, though the modeling of these aspects of the flow do show some room for improvement. The nonlinear ν^2 - f -cc turbulence model shows very non-physical skin friction coefficient profiles, pressure coefficient profiles, and stagnation points, indicating that the streamline curvature correction terms need attention. Regardless of the source of the discrepancies, the ν^2 - f turbulence models show promise in the modeling of circulation control flow fields, but are not quite ready for application in the design of circulation control aircraft.

TABLE OF CONTENTS

List of Tables	ix
List of Figures	x
Nomenclature	xiii
1. Introduction.....	1
1.1. Motivation and Objectives	1
1.2. Circulation Control Technology.....	2
1.3. Coanda Effect.....	3
1.4. Previous CFD Approaches to Circulation Control.....	3
1.4.a. Reynolds-Averaged Navier-Stokes Modeling	5
1.4.b. Large Eddy Simulation Modeling.....	14
1.4.c. Direct Numerical Simulation Modeling.....	17
2. Turbulence Modeling.....	18
2.1. Challenges to CFD Modeling of Circulation Control Flows	18
2.1.a. Streamline Curvature	18
2.1.b. Eddy Viscosity Formulation	20
2.2. Common Turbulence Models and their Shortcomings	22
2.2.a. Spalart-Allmaras	22
2.2.b. Spalart-Allmaras Model with Rotation and/or Curvature Effects (SARC)	25

2.2.c.	k- ϵ	26
2.2.d.	k- ω	28
2.3.	The v^2 - f Turbulence Models.....	32
2.3.a.	Standard v^2 - f Turbulence Model.....	32
2.3.b.	Nonlinear v^2 - f Turbulence Model.....	35
2.3.c.	Streamline Curvature Corrected Nonlinear v^2 - f Turbulence Model.....	40
2.3.d.	Calibration of Model Constants.....	46
3.	Use of User-Defined Functions with FLUENT.....	49
3.1.	User-Defined Functions, Scalars, and Memory.....	49
3.2.	Implementation of the v^2 - f Turbulence Models.....	51
3.2.a.	Source Code.....	51
3.2.b.	Coupling the Source Code to FLUENT.....	53
4.	Validation Cases.....	56
4.1.	Flat Plate in Turbulent, Subsonic Flow.....	56
4.2.	S3H4 2D Hill.....	59
5.	Circulation Control Airfoil Results (2D).....	62
5.1.	Previous CFD Results.....	62
5.2.	CFD Results with the v^2 - f Turbulence Models.....	62
6.	Conclusion.....	78
7.	Future Work.....	78

References	80

LIST OF TABLES

Table 2-1. Calibrated Constants for ν^2 - f Turbulence Models	49
Table 5-1. Boundary conditions for GACC airfoil CFD cases	64

LIST OF FIGURES

Figure 1-1: Circulation control airfoil with leading and trailing edge jets ³	2
Figure 1-2: Circulation control airfoil used by Pfingsten et al., with rounded trailing edge and computational mesh ¹	10
Figure 1-3: C_p distributions from Pfingsten et al. ¹⁸ distribution for $C_{\mu} = 0.1$ for a circulation control airfoil with rounded trailing edge	10
Figure 1-4: CFD and experimental results from Lee-Rausch et al. ¹¹ for the GACC airfoil at $M_{\infty} = 0.0824$ and $Re_c = 0.46 \times 10^6$	11
Figure 1-5: CFD and experimental results from Jones et al. ²⁰ for the GACC airfoil	12
Figure 1-6: CFD Results from Swanson et al. ¹²	13
Figure 1-7: Computational mesh near the NCCR 1510-7067N circulation control airfoil from Slomski et al. Top: showing outer boundary. Bottom left: showing near-airfoil region. Bottom right: showing trailing edge region.	15
Figure 1-8: Surface pressure distributions for a RANS solution (left) and a LES solution (right) from Slomski, et al. compared to experimental results.....	16
Figure 1-9: Profiles of turbulence kinetic energy (left) and mean velocity (right) for RANS and LES models at the trailing edge from Slomski et al.	17
Figure 2-1: Comparison of turbulence models from Heschl et al.	40
Figure 2-2: Eddy viscosity contours (normalized by molecular viscosity) at $x/c = 0.246$ for a wingtip vortex flow, from Duraisamy et al.	45
Figure 2-3: Vertical velocity (normalized by free-stream velocity) along a line passing through a wingtip vortex core, from Duraisamy et al.	46

Figure 2-4. Calibration curves: C_1 (top-left); C_2 (top-right); C_μ (middle-left); C_η (middle-right); C_L (bottom)	48
Figure 3-1: Architecture for user access to the FLUENT solver	51
Figure 4-1: Skin friction coefficient distribution for a turbulent flat plate in subsonic flow	57
Figure 4-2: v^2 - f boundary layer velocity profiles for a turbulent flat plate	58
Figure 4-3: Boundary layer velocity profiles for a turbulent flat plate	59
Figure 4-4: S3H4 2D hill velocity profiles (solid: experimental; line: CFD)	61
Figure 5-1: GACC airfoil	63
Figure 5-2: Computational grid for GACC airfoil: farfield (top-left); nearfield (top-right); leading edge (middle-left); flap (middle-right); circulation control slot (bottom)	66
Figure 5-3: CFD results for the v^2 - f turbulence model compared to common turbulence models and experimental data	68
Figure 5-4: Pressure coefficient distribution comparison for the GACC airfoil at $C_\mu=0.084$	70
Figure 5-5: Experimental stagnation point and jet velocity contours for the GACC airfoil at $C_\mu=0.084$ (Jones et al.)	71
Figure 5-6: Stagnation point and jet velocity comparison for the GACC airfoil at $C_\mu=0.084$. Top-left: Spalart-Allmaras; top-right: k - ϵ ; middle-left: k - ω ; middle-right: linear v^2 - f ; bottom-left: nonlinear v^2 - f ; bottom-right: nonlinear v^2 - f -cc	72
Figure 5-7: Experimental jet velocity contours for the GACC airfoil at $C_\mu=0.084$ (Jones et al.)	73

Figure 5-8: Jet velocity comparison for the GACC airfoil at $C_\mu=0.084$. Top-left: Spalart-Allmaras; top-right: $k-\epsilon$; middle-left: $k-\omega$; middle-right: linear v^2-f ; bottom-left: nonlinear v^2-f ; bottom-right: nonlinear v^2-f -cc	74
Figure 5-9: Skin friction coefficient distribution comparison for CFD results the GACC airfoil at $C_\mu=0.084$	76
Figure 5-10: Drag coefficient as a function of blowing coefficient for the GACC airfoil.....	77

NOMENCLATURE

Alphabetic

a	= speed of sound
$b_{ij}, \mathbf{b}$	= Reynolds stress anisotropy tensor
b	= reference length
C_1	= calibrated constant
$C_{1\epsilon}$	= calibrated constant
C_2	= calibrated constant
$C_{2\epsilon}$	= calibrated constant
$C_{3\epsilon}$	= calibrated constant
C_L	= calibrated constant
C_L	= three-dimensional lift coefficient
C_T	= calibrated constant
C_{b1}	= calibrated constant
C_{b2}	= calibrated constant
C_d	= sectional drag coefficient
CE_2	= calibrated constant
CE_D	= calibrated constant
C_f	= skin friction coefficient
C_l	= sectional lift coefficient
C_{r1}	= calibrated constant
C_{r2}	= calibrated constant
C_{r3}	= calibrated constant
C_{w1}	= calibrated constant
C_{w2}	= calibrated constant
C_{w3}	= calibrated constant
C_η	= calibrated constant
C_μ	= jet momentum coefficient
$C_{\epsilon 1}^*$	= calibrated constant
$C_{\epsilon 2}$	= calibrated constant
d	= distance from a wall
e	= energy
F	= user-defined flux function
f	= elliptic relaxation factor
f_1	= measure of streamline curvature
f_{r1}	= rotation function
f_{v1}	= viscous damping function
$G_{\tilde{\nu}}$	= production of turbulent viscosity
G_b	= buoyancy correction factor
G_k, P_k	= production of turbulence kinetic energy
G_ω	= production of specific dissipation rate
g^λ	= general calibrated expansion coefficient for a nonlinear eddy viscosity model

H	= hill height
h	= enthalpy
h	= reference height
$\mathbf{I}$	= identity matrix
k	= turbulence kinetic energy
L	= turbulence length scale
L_1	= hill half-length
M_t	= turbulent Mach number
$\dot{m}$	= mass flow rate
P	= total pressure
p	= pressure
Re	= Reynolds number
R_{ij}	= Reynolds stress tensor
S	= hill slope
S	= wing reference area
$S_{\bar{v}}$	= turbulent viscosity source
S_{Φ}	= user-defined source term
$S_{ij}, \mathbf{S}$	= mean strain rate tensor, $= \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$
$\mathbf{T}$	= transformation matrix
T	= total temperature
T	= turbulence time scale
T_{ij}^{λ}	= general tensor base for a nonlinear eddy viscosity model
U	= velocity
u, v, w	= Cartesian velocity components
u^+	= dimensionless wall velocity
v^2	= velocity scale
x, y, z	= Cartesian coordinates
$Y_{\bar{v}}$	= destruction of turbulent viscosity
Y_M	= compressibility correction factor
Y_k	= dissipation of turbulence kinetic energy
Y_{ω}	= dissipation of specific dissipation rate
y^+	= dimensionless wall distance

Greek

α	= angle of attack
α	= damping function
α^*	= damping function
β	= transport of kinetic energy
Γ	= user-defined diffusivity function
Γ_k	= effective diffusivity of turbulence kinetic energy
Γ_{ω}	= effective diffusivity of specific dissipation rate
δ_{ij}	= Kronecker delta, $= \begin{cases} 1, & i = j \\ 0, & i \neq j \end{cases}$
ε	= turbulence dissipation rate

ε_{ijk}	= cyclic permutation tensor, = $\begin{cases} 1, & ijk = 123, 231, 312 \\ -1, & ijk = 321, 213, 132 \\ 0, & \text{all other } ijk \end{cases}$
η_1	= velocity gradient invariant
η_2	= velocity gradient invariant
η_3	= velocity gradient invariant
κ	= von Karman constant, ≈ 0.41
κ	= thermal conductivity
λ	= second viscosity coefficient
μ	= molecular dynamic viscosity
μ_t	= turbulent dynamic viscosity
ν	= molecular kinematic viscosity, $= \frac{\mu}{\rho}$
$\tilde{v}$	= transported variable in the Spalart-Allmaras turbulence model
ν_t	= turbulent kinematic viscosity, $= \frac{\mu_t}{\rho}$
ρ	= density
σ_k	= turbulent Prandtl number for turbulence kinetic energy
σ_v	= calibrated constant
σ_ϵ	= turbulent Prandtl number for turbulence dissipation rate
τ	= turbulence time scale
τ_1	= calibrated constant
τ_{ij}	= turbulent shear stress
Φ	= user-defined variable
Ψ	= turbulent heat flux
ψ	= general flow field variable
$\Omega_{ij}, \mathbf{\Omega}$	= mean rotation rate tensor, $= \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right)$
$\bar{\Omega}_{ij}$	= objective vorticity tensor
ω	= specific dissipation rate
ω_i	= vorticity term

Superscripts, Subscripts and Other Symbols

0	= stagnation (total)
∞	= freestream
T	= transpose of a matrix
bc	= wall boundary condition
i, j, k	= Cartesian
jet	= circulation control jet

Acronyms

CC	= circulation control
cc	= curvature corrected
$CESTOL$	= cruise-efficient takeoff and landing
CFD	= computational fluid dynamics
DNS	= direct numerical simulation

GACC = general aviation circulation control
LES = large eddy simulation
LEVM = linear eddy viscosity model
NASA = National Aeronautics and Space Association
NLEVM = nonlinear eddy viscosity model
PIV = particle image velocimetry
RANS = Reynolds-averaged Navier-Stokes
SARC = Spalart-Allmaras model with rotation and/or curvature effects
SSARC = simplified Spalart-Allmaras model with rotation and/or curvature effects
UDF = user-defined function
UDM = user-defined memory
UDS = user-defined scalar

1. Introduction

1.1.Motivation and Objectives

In recent years, airports have experienced increasing airport congestion, partially due to the hub-and-spoke model on which airline operations are based. Current airline operations utilize large airports, focusing traffic to a small number of airports. One way to relieve such congestion is to transition to a more accessible and efficient point-to-point operation, which utilizes a large web of smaller airports. This expansion to regional airports propagates the need for next-generation low-noise aircraft with short take-off and landing capabilities. NASA has attacked this problem with a high-lift, low-noise concept dubbed the Cruise Efficient Short Take-Off and Landing (CESTOL) aircraft. The ultimate goal of the CESTOL project is to produce aircraft designs that can further expand the air travel industry to currently untapped regional airports.

The primary challenges in accurately modeling CESTOL aircraft performance include the ability to capture propulsion and aerodynamic coupling, to calculate balanced field length, and to correctly model the aerodynamics. Also, the CESTOL aircraft concept necessitates a balance between cruise efficiency and short takeoff and landing, which are generally opposing forces in aircraft design. The unique configurations of CESTOL aircraft pose a significant CFD challenge, motivating the improvements to turbulence modeling outlined in this thesis.

1.2.Circulation Control Technology

To generate increased lift from traditional subsonic airfoils, either the angle of attack or the camber must be increased. The maximum lift coefficient of a traditional wing is limited by the eventual separation of flow over the wing, due primarily to the adverse pressure gradient that builds on the wing as lift is increased. Traditionally, this obstacle is overcome by use of complex moving wing surfaces, including flaps, slats, and other devices.

Circulation control has been proposed as a simpler and more effective alternative to the usual high-lift devices¹. Circulation control is an active flow control device that increases the lift coefficient without the use of complex components in freestream flow. Circulation control is primarily needed when high lift coefficients are required due to low airspeeds, particularly during takeoff and landing. The technology makes use of the Coanda effect, according to which a fluid has a tendency to stay attached to an adjacent curved surface². A high-speed jet of air is blown out of the leading and/or trailing edge of a wing, which follows the wing surface. The stagnation point on the leading edge and the flow separation point on the trailing edge are thus manipulated such that the circulation is increased, and consequently lift is increased.

Figure 1-1: Circulation control airfoil with leading and trailing edge jets³

The extent of the stagnation and separation point movement is primarily a function of the jet momentum coefficient, C_μ . The jet momentum coefficient is a measure of the jet momentum relative to the freestream momentum, and has two common formulations, which are defined as follows⁴.

$$C_\mu = \frac{2h_{jet} b_{jet}}{S} \frac{\rho_{jet}}{\rho_\infty} \frac{U_{jet}^2}{U_\infty^2} = \frac{\dot{m}U_{jet}}{q_\infty S} \quad (1.1)$$

The benefit of circulation control is that the lift is significantly increased, with relatively insignificant increases in drag. Circulation control requires high energy air to be blown over the wing's surface. This requires some additional source of energy, potentially the engine(s) or auxiliary gas generators; this is, of course, a problem associated with circulation control, but the solution is outside the scope of this research paper.

1.3.Coanda Effect

Circulation control technology is based on the concept of the Coanda effect, which is the tendency of high-speed, pressurized jets to deflect toward nearby solid surfaces⁵. The Coanda effect is named for the Romanian researcher Henri Coanda, who noticed that the hot engine exhaust on his Coanda-1910 jet-propelled aircraft tended to hug the aircraft's fuselage.

1.4.Previous CFD Approaches to Circulation Control

Traditional CFD approaches have been applied to circulation control airfoils, mostly in two dimensions, with mixed success. Unfortunately, the physics of circulation control

wings are highly complex, and are not well understood due to limited experimental studies. The high momentum of the circulation control jet allows the boundary layer to remain attached longer than usual, thereby moving the separation point. This movement of the separation point is the primary reason that lift is augmented, and any CFD modeling techniques must be able to accurately model the separation point by properly predicting the spreading rate of the jet and the exchange of momentum between the jet and the surrounding fluid.

Common numerical methods all have attributes that limit their use in circulation control applications, these methods including Direct Numerical Simulation (DNS), Large Eddy Simulation (LES), and Reynolds-Averaged Navier-Stokes (RANS). Today's computer resources limit most academic and industrial CFD to RANS solutions, especially for high Reynolds number flows. Many attempts have been made to model circulation control flow fields using common turbulence models (including Baldwin-Lomax^{6,7,8}, Spalart-Allmaras^{1,7,8,9,10,11,12}, $k-\epsilon$ ¹³, and $k-\omega$ ^{11,12,13,14}), and while acceptable accuracy has been obtained in some cases, the general consensus has been that these models are not well suited for circulation control flow fields.

The following sections summarize previous studies using CFD to analyze circulation control airfoils, and show that the common approaches to modeling these flow fields are inappropriate and provide only marginally accurate results. Clearly, a tool needs to be adapted such that predictions for circulation control flow fields are greatly improved, to the point where CFD can be used as the primary tool for designing an aircraft with circulation control.

1.4.a. Reynolds-Averaged Navier-Stokes Modeling

RANS models are solutions to the time-averaged equations of motion for fluid flow. These time-averaged equations of motion can be used to provide approximate time-averaged solutions to the full Navier-Stokes equations. The general process for deriving the Reynolds-Averaged Navier-Stokes equations is to first replace flow field variables with Favre-averaged properties, and to time-average the resulting equations. To begin, a flow field variable ψ can be generalized as being the sum of an time-averaged value and a fluctuating value, where $\psi = \tilde{\psi} + \psi''$. For this general Favre-averaged variable, the generalized continuity equation is

$$\begin{aligned} \frac{\partial}{\partial t}(\rho\psi) + \frac{\partial}{\partial x_i}(\rho\psi u_i) \\ = \frac{\partial}{\partial t}(\overline{\rho\tilde{\psi}} + \overline{\rho\psi''}) \\ + \frac{\partial}{\partial x_i}[\overline{\rho(\tilde{u}_i\tilde{\psi} + \tilde{u}_i\psi'' + u_i''\tilde{\psi} + u_i''\psi'')}] \end{aligned} \quad (1.2)$$

According to the principal of Favre averaging, $\overline{\rho\psi''} = 0$, and Equation (1.2) reduces to

$$\begin{aligned} \frac{\partial}{\partial t}(\rho\psi) + \frac{\partial}{\partial x_i}(\rho\psi u_i) \\ = \frac{\partial}{\partial t}(\bar{\rho}\tilde{\psi}) + \frac{\partial}{\partial x_i}(\bar{\rho}\tilde{u}_i\tilde{\psi}) + \frac{\partial}{\partial x_i}(\overline{\rho u_i''\psi''}) \end{aligned} \quad (1.3)$$

When $\psi = 1$, $\psi'' = 0$, and Equation (1.3) reduces to the RANS form of the continuity equation:

$$0 = \frac{\partial}{\partial t}(\bar{\rho}) + \frac{\partial}{\partial x_i}(\bar{\rho}\tilde{u}_i) \quad (1.4)$$

The momentum equation is

$$\frac{\partial}{\partial t}(\rho u_j) + \frac{\partial}{\partial x_i}(\rho u_i) + \delta_{ij} \frac{\partial P}{\partial x_i} = \frac{\partial \tau_{ij}}{\partial x_i} \quad (1.5)$$

where

$$\tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \lambda \frac{\partial u_k}{\partial x_k} \quad (1.6)$$

and $\lambda = -\frac{2}{3}\mu$ according to Stoke's hypothesis.

From Equation (1.3), if $\psi = u_j$ and $\psi'' = u_j''$, the unsteady and convective terms of the momentum equation are

$$\overline{\frac{\partial}{\partial t}(\rho u_j) + \frac{\partial}{\partial x_i}(\rho u_i u_j)} = \frac{\partial}{\partial t}(\bar{\rho} \tilde{u}_j) + \frac{\partial}{\partial x_i}(\bar{\rho} \tilde{u}_i \tilde{u}_j) - \frac{\partial R_{ij}}{\partial x_i} \quad (1.7)$$

where $R_{ij} = -\overline{\rho u_i'' u_j''}$ is the Reynolds stress tensor.

Assuming that $\mu' \approx 0$, $\lambda' \approx 0$, and $\frac{\partial \overline{u_i''}}{\partial x_j} \approx 0$, the pressure and viscous stress terms of the momentum equation are as follows.

$$\overline{\delta_{ij} \frac{\partial P}{\partial x_i}} = \delta_{ij} \frac{\partial \bar{P}}{\partial x_i} \quad (1.8)$$

$$\bar{\tau}_{ij} = 2\bar{\mu} \tilde{S}_{ij} + \bar{\lambda} \frac{\partial \bar{u}_k}{\partial x_k} \quad (1.9)$$

where

$$\tilde{S}_{ij} = \frac{1}{2} \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) \quad (1.10)$$

The unsteady, convective, pressure, and viscous stress terms for the momentum equation can be combined to yield the following RANS form of the momentum equation.

$$\frac{\partial}{\partial t}(\bar{\rho}\tilde{u}_j) + \frac{\partial}{\partial x_i}(\bar{\rho}\tilde{u}_i\tilde{u}_j) + \delta_{ij}\frac{\partial\bar{P}}{\partial x_i} - \frac{\partial}{\partial x_i}(\bar{\tau}_{ij} + R_{ij}) = 0 \quad (1.11)$$

This same process can be applied to the energy equation, which is

$$\frac{\partial(\rho e_0)}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i h_0) = \frac{\partial}{\partial x_i}(u_i \tau_{ij}) - \frac{\partial q_j}{\partial x_j} \quad (1.12)$$

where $q_j = -\kappa \frac{\partial T}{\partial x_j}$. The unsteady and convective terms are Favre-averaged in the same

manner as for the continuity and momentum equations, resulting in the following:

$$\begin{aligned} \frac{\partial}{\partial t}(\rho e_0) + \frac{\partial}{\partial x_i}(\rho u_i h_0) \\ = \frac{\partial}{\partial t}(\bar{\rho}\tilde{e}_0) + \frac{\partial}{\partial x_i}(\bar{\rho}\tilde{u}_i\tilde{h}_0) + \frac{\partial\Psi_i}{\partial x_i} \\ - \frac{\partial}{\partial x_i}(\tilde{u}_j R_{ij}) - \frac{\partial\beta_i}{\partial x_i} \end{aligned} \quad (1.13)$$

where $\Psi_i = \overline{\rho u_i'' h''}$ is the turbulent heat flux, $\beta_i = -\frac{1}{2}\overline{\rho u_i'' u_j'' u_j''}$ is the transport of

turbulence kinetic energy, $k = \frac{\overline{\rho u_i'' u_i''}}{2\rho}$ is the turbulence kinetic energy, $\tilde{e}_0 = \tilde{e} + \frac{1}{2}\tilde{u}_i\tilde{u}_i +$

k is the total energy, $\tilde{h}_0 = \tilde{e}_0 + \frac{\bar{P}}{\rho}$ is the total enthalpy, and κ is the thermal conductivity.

The transport of turbulence kinetic energy only becomes significant in hypersonic flows, and is assumed zero for subsonic flows¹⁵. The Favre-averaging of the conduction term

requires the assumptions that $\kappa' \approx 0$ and $\frac{\partial T''}{\partial x_i} \approx 0$, and results in the following:

$$\bar{q}_i = -\bar{\kappa} \frac{\partial\tilde{T}}{\partial x_i} \quad (1.14)$$

The Favre-averaging of the viscous term results in the following:

$$\overline{u_j \tau_{ij}} = (\tilde{u}_j + \overline{u_j''}) \bar{\tau}_{ij} \quad (1.15)$$

Finally, combining the unsteady, convective, conduction, and viscous terms results in the following RANS form of the energy equation.

$$\begin{aligned} \frac{\partial}{\partial t}(\bar{\rho} \tilde{e}_0) + \frac{\partial}{\partial x_i}(\bar{\rho} \tilde{u}_i \tilde{h}_0) - \frac{\partial}{\partial x_i}[\tilde{u}_j (\bar{\tau}_{ij} + R_{ij}) + \beta_i] \\ + \frac{\partial}{\partial x_i}(\bar{q}_i + \Psi_i) = 0 \end{aligned} \quad (1.16)$$

where $\tilde{e}_0 = \tilde{e} + \frac{1}{2} \tilde{u}_i \tilde{u}_i + k$, $\tilde{h}_0 = \tilde{e}_0 + \frac{\bar{p}}{\bar{\rho}}$, and $k = \frac{\overline{\rho u_i'' u_j''}}{2\bar{\rho}}$.

The Boussinesq approximation simplifies the approximation of the Reynolds stresses, and is applied to alter Equations (1.11) and (1.12). According to the Boussinesq approximation, the Reynolds stresses are formulated as follows.

$$R_{ij} = 2\mu_t \tilde{S}_{ij} + \lambda_t \frac{\partial \tilde{u}_k}{\partial x_k} - \frac{2}{3} \delta_{ij} \bar{\rho} k \quad (1.17)$$

The sum of the viscous stress and Reynolds stresses is reformulated using the Boussinesq assumption, resulting in the following.

$$\tau_{ij}^* = \bar{\tau}_{ij} + R_{ij} = 2(\bar{\mu} + \mu_t) \tilde{S}_{ij} + (\bar{\lambda} + \lambda_t) \frac{\partial \tilde{u}_k}{\partial x_k} - \frac{2}{3} \delta_{ij} \bar{\rho} k \quad (1.18)$$

This formulation is used in Equations (1.11) and (1.16) (the momentum and energy equations). Additionally, it can be shown that the conduction term $\bar{q}_i$ can be expressed as

$$q_i^* = -(\bar{k} + k_t) \frac{\partial \tilde{T}}{\partial x_i} \quad (1.19)$$

where

$$k_t = \frac{C_p \mu_t}{Pr_t} \quad (1.20)$$

Applying the Boussinesq approximation to the RANS equations results in the following forms of the continuity, momentum, and energy equations.

$$\begin{aligned} \frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_i} (\bar{\rho} \tilde{u}_i) &= 0 \\ \frac{\partial}{\partial t} (\bar{\rho} \tilde{u}_j) + \frac{\partial}{\partial x_i} (\bar{\rho} \tilde{u}_i \tilde{u}_j) + \delta_{ij} \frac{\partial \bar{P}}{\partial x_i} - \frac{\partial \tau_{ij}^*}{\partial x_i} &= 0 \\ \frac{\partial}{\partial t} (\bar{\rho} \tilde{e}_0) + \frac{\partial}{\partial x_i} (\bar{\rho} \tilde{u}_i \tilde{h}_0) - \frac{\partial}{\partial x_i} \left[\tilde{u}_j \tau_{ij}^* + \left(\bar{\mu} + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + \frac{\partial q_i^*}{\partial x_i} &= 0 \end{aligned} \quad (1.21)$$

The process of time-averaging eliminates the need to fully resolve all scales of turbulent motion, but introduces the Reynolds stresses, which are interpreted as quantities associated with turbulent behavior. These quantities are not directly known, and must be related to the mean flow via turbulence closure models (also commonly referred to as eddy viscosity closure models). Common examples of turbulence closure models are the Spalart-Allmaras, k - ϵ , and k - ω models; details of these models are presented in the following section.

Pfingsten et al.¹⁸ performed numerical two-dimensional simulations with a RANS solver for a circulation control airfoil with a rounded trailing edge, and compared the results to experimental data collected by Novak et al.¹⁹. Turbulence closure was achieved using the Spalart-Allmaras turbulence model and two variations thereof; the two variations are the Spalart-Allmaras model with Rotation and/or Curvature effects (SARC,) and the Simplified Spalart-Allmaras model with Rotation and/or Curvature effects (SSARC,) both of which account for streamline curvature effects. While the

streamline curvature correction improved predictions, all three models consistently under-predicted the surface pressure coefficient, and consequently over-predicted integrated force coefficients by 7-32%. Furthermore, the results presented were for relatively low jet momentum coefficients, and it is possible that these turbulence models would perform more poorly for higher jet momentum coefficients.

Figure 1-2: Circulation control airfoil used by Pfingsten et al., with rounded trailing edge and computational mesh¹

Figure 1-3: C_p distributions from Pfingsten et al.¹⁸ distribution for $C_\mu = 0.1$ for a circulation control airfoil with rounded trailing edge

Lee-Rausch et al.¹¹ presented computational results for the GACC airfoil using two RANS solvers (CFL3D and Fun3D,) and compared the results to experimental data collected in the LaRC BART wind tunnel. Both solvers used the standard Spalart-Allmaras turbulence model, and the results show the same over-prediction of integrated coefficients.

Figure 1-4: CFD and experimental results from Lee-Rausch et al.¹¹ for the GACC airfoil at $M_{\infty} = 0.0824$ and $Re_c = 0.46 \times 10^6$

Jones et al.²⁰ conducted a similar study using the GACC airfoil using the SARC and Menter's $k-\omega$ - SST turbulence models, and concluded that the poor match between CFD performance predictions and experimental data was due to the turbulence models as well as grid issues. The authors managed to match experimental stagnation point location and pressure coefficients quite well by adjusting the angle of attack of the CFD model, but the

ultimate goal is to avoid such ad-hoc approaches and develop a modeling technique that captures circulation control flow phenomena without these adjustments.

Figure 1-5: CFD and experimental results from Jones et al.²⁰ for the GACC airfoil

McGowan et al.^{9,21} also studied the GACC airfoil using the Spalart-Allmaras turbulence model. The results capture the trends associated with circulation control, specifically the movement of the stagnation point and the increase in lift related to increasing jet momentum coefficient. However, the authors acknowledge that their quantitative results, such as lift coefficient, do not exactly match experimental data.

Swanson et al.¹² used the RANS solver CFL3D with the Spalart-Allmaras, SARC, Menter's $k-\omega-SST$, and $k-\zeta$ turbulence models, and results were compared to experimental data collected by Novak et al.¹⁹ The results indicate that streamline curvature correction is a necessary component of turbulence models as applied to circulation control, as the

SARC model performs better than the other turbulence models. However, the SARC model still over-predicted the lift coefficient by 10-29%.

Figure 1-6: CFD Results from Swanson et al.¹²

In his Master of Science thesis, Liu⁷ studied numerical simulations of circulation control wing sections using the Baldwin-Lomax and Spalart-Allmaras turbulence models. He concluded that the models perform reasonably well when the flow is fully attached and there is no separation, but their performance quickly degrades when these phenomena occur. Furthermore, these models did not accurately simulate the strong tip vortices that circulation control wings generate. In his conclusion, Liu suggests that advanced turbulence models be studied with circulation control in mind.

One significant observation to make about all of these circulation control studies is that the turbulence models used are all linear eddy viscosity models that make use of the

Boussinesq eddy viscosity hypothesis. The Boussinesq assumption simplifies the relationship between the turbulent Reynolds stresses and the mean strain rate tensor to a linear relationship, the scalar multiple being the isotropic eddy viscosity. Also, when streamline curvature effects were incorporated into the Spalart-Allmaras model, results were universally improved, though the base model does not provide proper robustness for circulation control flows. Later sections will show the derivations of nonlinear eddy viscosity formulations and streamline curvature correction for the v^2 - f turbulence model.

1.4.b. Large Eddy Simulation Modeling

LES is a simplification of DNS, in which the large-scale solution is explicitly solved in a manner similar to DNS, but the small-scale solution is modeled in a manner similar to RANS solutions. In a LES solution, spatially-averaged Navier-Stokes equations (unlike the time-averaged Navier-Stokes equations used for RANS solutions) are filtered, to determine which scales are to be solved directly and which are to be modeled. LES simulations are processor and memory intensive, and it is unlikely that such simulations would be practical in complex three-dimensional cases. However, simple LES solutions can provide a strong basis of comparison and validation for other modeling techniques, particularly in two dimensions.

Slomski et al.²² generated a LES solution for the NCCR 1510-7067N airfoil, a circulation control airfoil with a rounded trailing edge. The simulation required nearly 19 million mesh points, supporting the claim that LES is only practical for two-dimensional circulation control airfoils, and application to three-dimensional wings or full aircraft is beyond the capabilities of most computers. Slomski et al. used the CRAFT solver to solve

the filtered form of the Navier-Stokes equations; for the sake of brevity, the governing equations will not be reproduced in this paper. The airfoil and structured grid are shown in the following figures.

Figure 1-7: Computational mesh near the NCCR 1510-7067N circulation control airfoil from Slomski et al. Top: showing outer boundary. Bottom left: showing near-airfoil region. Bottom right: showing trailing edge region.

A RANS solution was used to initialize the LES solver, and the results from the two solutions were compared. The two solutions showed very similar surface pressure distributions (see Figure 1-8,) though the RANS solution appears to better match experimental data on the aft half of the lower surface.

Figure 1-8: Surface pressure distributions for a RANS solution (left) and a LES solution (right) from Slomski, et al. compared to experimental results

However, the results differed significantly for turbulence kinetic energy and mean velocity on the rounded trailing edge. The LES model predicted significantly higher turbulence kinetic energy and mean velocity than did the RANS model, as can be seen in Figure 1-9.

Figure 1-9: Profiles of turbulence kinetic energy (left) and mean velocity (right) for RANS and LES models at the trailing edge from Slomski et al.

For these quantities, Slomski et al. conclude that the RANS solution must be able to sufficiently model diffusion of momentum and turbulence kinetic energy from the circulation control jet. The authors note significant anisotropy of the Reynolds stresses in the shear layer in the LES solution. The failure of the RANS solution to correctly model turbulence kinetic energy and mean velocity in the circulation control jet can be blamed, at least in part, on the Boussinesq assumption used in the turbulence model. This indicates that a nonlinear eddy viscosity model may improve results by capturing anisotropy of the Reynolds stresses.

1.4.c. Direct Numerical Simulation Modeling

DNS is a technique in which the Navier-Stokes equations are solved without any turbulence modeling, requiring that the complete range of spatial and temporal scales of turbulence be resolved. This requires a grid fine enough to resolve the flow field down to the smallest turbulence scales (the length, time, and velocity Kolmogorov microscales.)

and as such would prove a powerful tool for both predicting circulation control performance as well as validating other modeling techniques. However, DNS requires an explicit solution to the governing equations, which implies that the timestep must be small enough such that fluid particles do not move greater than one cell size (i.e., the Courant number must be less than 1). This, along with the memory requirements of the very fine grid, creates memory and processing demands for flows with high Reynolds number flows (including circulation control flows) that greatly exceed the available resources of even the most powerful computers. To date, no DNS solution has been produced for circulation control flows.

2. Turbulence Modeling

2.1.Challenges to CFD Modeling of Circulation Control Flows

2.1.a. Streamline Curvature

In the formulation of an algebraic stress model, the classical approach is to apply the weak-equilibrium assumption. For the turbulence equations, the weak-equilibrium assumption forces the Reynolds stress anisotropy tensor $b_{ij} = \frac{\overline{\rho u_i'' u_j''}}{\bar{\rho} k} - \frac{2}{3} \delta_{ij}$ to be constant along a streamline²³. This assumption is untrue in flows with streamline curvature, and the effect is amplified in cases where such curvature is significant.

The calculation of material derivatives is necessary to solve the RANS equations. Material derivatives of scalar fields are Galilean invariant (i.e., invariant of the coordinate system;) however, material derivatives of tensor fields with rank greater than zero are not Galilean invariant²³. The material derivative of the Reynolds stress anisotropy tensor is

needed to solve the RANS equations, presenting a problem in flows with streamline curvature. Spalart and Shur²⁴ proposed a method based on a Galilean invariant measure of turbulence for sensitizing eddy viscosity turbulence models to the effects of streamline curvature. Gatski et al.²⁵ and Hellsten et al.²⁶ proposed extensions of this approach, in which the anisotropy tensor is transformed to a local streamline-oriented coordinate system such that the weak equilibrium assumption is valid. The general model for the transport of the anisotropy tensor is given as follows. For the sake of simplicity, Favre averaging is assumed, and the tilde accents ($\tilde{}$) are not shown.

$$\begin{aligned} \frac{D\mathbf{b}}{Dt} = & -\frac{\mathbf{b}}{a_4} - a_3 \left(\mathbf{b}\mathbf{S} + \mathbf{S}\mathbf{b} - \frac{2}{3}\{\mathbf{b}\mathbf{S}\}\mathbf{I} \right) + a_2(\mathbf{b}\mathbf{W} - \mathbf{W}\mathbf{b}) \\ & + \frac{a_5\varepsilon}{k} \left(\mathbf{b}^2 - \frac{1}{3}\{\mathbf{b}^2\}\mathbf{I} \right) \end{aligned} \quad (2.1)$$

where a_i are calibrated constants and $\{ \}$ represents the trace of a matrix. The weak equilibrium assumption of this form gives $\frac{D\mathbf{b}}{Dt} = 0$, and leads to an algebraic system of equations for the Reynolds stress anisotropy tensor that is dependent on the choice of coordinate system. This can be corrected by introducing the transformation matrix $\mathbf{T}$, which accounts for the transformation from a global, inertial coordinate system to a local, streamline-oriented coordinate system. Hellsten et al. showed that taking the material derivative of $\mathbf{T}\mathbf{b}\mathbf{T}^T$ and transforming back into the inertial coordinate system results in:

$$\frac{D\mathbf{b}}{Dt} = \mathbf{T}^T \left[\frac{D}{Dt} (\mathbf{T}\mathbf{b}\mathbf{T}^T) \right] \mathbf{T} - (\mathbf{b}\mathbf{W} - \mathbf{W}\mathbf{b}) \quad (2.2)$$

where $\bar{\boldsymbol{\Omega}} = \mathbf{T} \frac{D\mathbf{T}^T}{Dt}$. The transformation matrix must be selected such that the Reynolds stress anisotropy can be ignored in the local, streamline-oriented coordinate system. If

this is true, the Reynolds stress anisotropy tensor transport equation simplifies to the following:

$$\frac{D\mathbf{b}}{Dt} = -(\mathbf{b}\bar{\boldsymbol{\Omega}} - \bar{\boldsymbol{\Omega}}\mathbf{b}) \quad (2.3)$$

This result can be incorporated into the original transport equation for the Reynolds stress anisotropy tensor by replacing $\mathbf{W}$ in Equation (2.1) with $\mathbf{W}^* = \frac{1}{a_2}\bar{\boldsymbol{\Omega}}$, resulting in the following, final form of the transport equation:

$$\begin{aligned} \frac{D\mathbf{b}}{Dt} = & -\frac{\mathbf{b}}{a_4} - a_3 \left(\mathbf{b}\mathbf{S} + \mathbf{S}\mathbf{b} - \frac{2}{3}\{\mathbf{b}\mathbf{S}\}\mathbf{I} \right) \\ & + a_2 \left[\mathbf{b} \left(\mathbf{W} + \frac{1}{a_2}\bar{\boldsymbol{\Omega}} \right) - \left(\mathbf{W} + \frac{1}{a_2}\bar{\boldsymbol{\Omega}} \right) \mathbf{b} \right] \\ & + \frac{a_5\varepsilon}{k} \left(\mathbf{b}^2 - \frac{1}{3}\{\mathbf{b}^2\}\mathbf{I} \right) \end{aligned} \quad (2.4)$$

The trouble in incorporating streamline curvature effects reduces to finding the transformation matrix, $\mathbf{T}$. For the case of the v^2 - f turbulence model, the derivation of the curvature-corrected model is shown on Page 40.

2.1.b. Eddy Viscosity Formulation

The traditional approach to relating the turbulent Reynolds stresses to the mean strain rate tensor has been to make use of the Boussinesq assumption, thereby creating a linear eddy viscosity model (LEVIM). According to the Boussinesq assumption, the unnormalized Reynolds stresses and the mean strain rate tensor are related via the turbulent viscosity such that²⁷

$$\overline{\rho u_i'' u_j''} = \frac{2}{3}\bar{\rho}k\delta_{ij} - 2\mu_t S_{ij} \quad (2.5)$$

where $S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$ is the mean strain rate tensor.

LEVM formulations vary in complexity from zero-equation (algebraic) to four-equation; the number of equations refers to the number of differential equations that need to be solved in a given model. Examples of zero-equation LEVMs include the Cebici-Smith²⁸ and Baldwin-Lomax²⁹ models. The most common one-equation LEVM is the Spalart-Allmaras turbulence model³⁰. Two-equation LEVMs include the k - ε ³¹ and k - ω ¹⁵ turbulence models. The standard v^2 - f turbulence model³² is an increasingly common four-equation turbulence model.

LEVMs have proven to be quite powerful in industrial and academic CFD applications, but the linearization of the relationship between the Reynolds stresses and the strain rate can cause these models to produce non-physical results. In particular, the Boussinesq assumption assumes that the eddy viscosity is isotropic; that is,

$$\overline{\rho u''^2} = \overline{\rho v''^2} = \overline{\rho w''^2} = \frac{2}{3} \bar{\rho} k \quad (2.6)$$

This negates any anisotropy of the Reynolds stresses, most notably near-wall anisotropy, which can be a significant feature of wall-bounded flows. While improper modeling of the normal stresses does not impact shear forces, capturing the anisotropy of the Reynolds stresses may be critical in capturing the wake region of circulation control flows.

One approach to solving this shortcoming has been to introduce empirical damping functions or other sorts of ad-hoc modifications. This allows improvement upon a model for a given type of flow, but is far from universal. Pope³³ suggested that the more robust

approach to this problem is to reformulate the relationship between the Reynolds stresses and the strain rate in a nonlinear manner. The general approach to formulating a nonlinear eddy viscosity model (NLEVM) is to generalize the formulation of the unnormalized Reynolds stresses to the following form¹⁵:

$$\overline{\rho u_i'' u_j''} = \frac{2}{3} \bar{\rho} k \delta_{ij} + \bar{\rho} k \sum_{\lambda} g^{\lambda} T_{ij}^{\lambda} \quad (2.7)$$

where T_{ij}^{λ} are the tensor bases and g^{λ} are the calibrated expansion coefficients. The specific approach to deriving the general form of the Reynolds stresses can vary depending on the number and form of the terms chosen to include in the tensor bases. One particular approach for the nonlinearization of the v^2 - f turbulence model is presented on Page 32.

2.2. Common Turbulence Models and their Shortcomings

2.2.a. Spalart-Allmaras

The Spalart-Allmaras turbulence model is a one-equation model that was presented by Spalart and Allmaras in 1992²². The model was motivated primarily by three things. First, the limited applicability of zero-equation models, such as the Baldwin-Lomax turbulence model, left a desire for a more robust turbulence modeling technique. Second, standard two-equation models, such as the k - ε turbulence model, involve strong source terms that often delay convergence, and a quicker method to a solution became desirable. Third, two-equation models typically require fine boundary layer meshes (often with the first cell inside the viscous sublayer,) and there arose a desire for looser meshing requirements. The Spalart-Allmaras turbulence model was calibrated specifically for

aerospace applications, in which boundary layers are subjected to adverse pressure gradients.

In the Spalart-Allmaras turbulence model, a single partial differential transport equation is solved at each timestep. The transported variable in this model is $\tilde{\nu}$, which is similar to the turbulent eddy viscosity (and is not a Favre averaged variable). This variable is transported according to the following transport equation³⁴.

$$\begin{aligned}
& \frac{\partial}{\partial t}(\rho\tilde{\nu}) + \frac{\partial}{\partial x_i}(\rho\tilde{\nu}u_i) \\
& = G_{\tilde{\nu}} \\
& + \frac{1}{\sigma_v} \left[\frac{\partial}{\partial x_i} \left\{ (\mu + \rho\tilde{\nu}) \frac{\partial}{\partial x_j} \right\} + C_{b2}\rho \left(\frac{\partial \tilde{\nu}}{\partial x_j} \right)^2 \right] \\
& - Y_{\tilde{\nu}} + S_{\tilde{\nu}}
\end{aligned} \tag{2.8}$$

In the above equation, $G_{\tilde{\nu}}$ is the production of turbulent viscosity, $Y_{\tilde{\nu}}$ is the destruction of turbulent viscosity, $S_{\tilde{\nu}}$ is a source term, ν is the molecular kinematic viscosity, and σ_v and C_{b2} are calibrated constants. The turbulent viscosity, μ_t , is computed as

$$\mu_t = \rho\tilde{\nu}f_{v1} \tag{2.9}$$

where f_{v1} is a viscous damping function defined as

$$f_{v1} = \frac{\left(\frac{\tilde{\nu}}{\nu}\right)^3}{\left(\frac{\tilde{\nu}}{\nu}\right)^3 + C_{v1}^3} \tag{2.10}$$

The production of turbulent viscosity is modeled as

$$G_v = C_{b1}\rho \left[S + \frac{\tilde{v}}{\kappa^2 d^2} \left(1 - \frac{\frac{\tilde{v}}{v}}{1 + \frac{\tilde{v}}{v} f_{v1}} \right) \right] \tilde{v} \quad (2.11)$$

where d is the distance from the wall, S is a scalar measure of the deformation tensor, C_{b1} is a calibrated constant, and κ is the von Karman constant ($\kappa \approx 0.41$). In the original Spalart-Allmaras model, S was based on the magnitude of the vorticity, such that

$$S = \sqrt{2\Omega_{ij}\Omega_{ij}} \quad (2.12)$$

where Ω_{ij} is the mean rate-of-rotation tensor,

$$\Omega_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \quad (2.13)$$

The formulation for S can vary, but the authors of the Spalart-Allmaras turbulence model used the mean rate-of-rotation tensor. This was justified by claiming that, for wall bounded flows, turbulence is found only where vorticity is generated near walls.

The turbulent destruction term, Y_v , is modeled as

$$Y_v = C_{w1}\rho f_w \left(\frac{\tilde{v}}{d} \right)^2 \quad (2.14)$$

where

$$f_w = g \left[\frac{1 + C_{w3}^6}{g^6 + C_{w3}^6} \right]^{1/6} \quad (2.15)$$

$$g = r + C_{w2}(r^6 - r) \quad (2.16)$$

$$r = \frac{\tilde{v}}{\left(S + \frac{\tilde{v}}{\kappa^2 d^2} f_{v2} \right) \kappa^2 d^2} \quad (2.17)$$

and C_{w1} , C_{w2} , and C_{w3} are calibrated constants.

The Spalart-Allmaras turbulence model has been very effective at modeling simple attached flows with no separation, but the Boussinesq assumption and insensitivity to streamline curvature and transition to turbulent flow limit the model's accuracy in complex circulation control flows. The Spalart-Allmaras model has been shown to over-predict integrated coefficients such as lift coefficient by more than 30% in some cases, and is not a reliable turbulence model for circulation control applications. Wilcox³⁵ shows that the Spalart-Allmaras model under-predicts spreading rates for jets, and concluded that the Spalart-Allmaras model is not appropriate for flows with jet-like free-shear regions (similar to those found in the wake of a circulation control wing).

2.2.b. *Spalart-Allmaras Model with Rotation and/or Curvature Effects (SARC)*

Shur et al.³⁶ added streamline curvature correction to the Spalart-Allmaras turbulence model, and the result has been applied numerous times to circulation control flows. The equations for the SARC turbulence model are the same as those for the standard Spalart-Allmaras turbulence model, except that the production term is multiplied by a rotation function, f_{r1} .

$$f_{r1} = (1 + C_{r1}) \frac{2r^*}{1 + r^*} [1 - C_{r3} \tan^{-1}(C_{r2} \tilde{r})] - C_{r1} \quad (2.18)$$

where

$$r^* = \frac{S}{\Omega} \quad (2.19)$$

$$\tilde{r} = \frac{2\Omega_{ik} S_{jk}}{D^4} \left(\frac{DS_{ij}}{Dt} \right) \quad (2.20)$$

$$S^2 = 2S_{ij} S_{ij} \quad (2.21)$$

$$\Omega^2 = 2\Omega_{ij}\Omega_{ij} \quad (2.22)$$

$$D^2 = \frac{1}{2}(S^2 + \Omega^2) \quad (2.23)$$

and C_{r1} , C_{r2} , and C_{r3} are calibrated constants.

The results produced for circulation control flows using the SARC model have been more physically accurate than those produced by the base Spalart-Allmaras turbulence model^{1,12,20}. However, the limitations associated with the base model still hold (e.g., the SARC model still cannot capture realistic spreading rates associated with jets, and the Boussinesq assumption limits application to turbulent flows,) and the model cannot correctly capture stagnation point movement or integrated coefficients for circulation control flows.

2.2.c. k - ε

The k - ε turbulence model is a two-equation model, proposed by Launder and Spalding³⁷. The model introduces transport equations for turbulence kinetic energy, which determines the energy in a turbulent flow, and turbulence dissipation rate, which determines the scale of the turbulence. The transport equation for k was derived exactly, but the transport equation for ε was derived according to physical reasoning and dimensional analysis. In this turbulence model, k and ε are determined according to the following two transport equations, respectively³⁸.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \varepsilon - Y_M \quad (2.24)$$

$$\begin{aligned}
& \frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) \\
&= \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) \\
&\quad - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k}
\end{aligned} \tag{2.25}$$

In the above equations, G_k accounts for the generation of turbulence kinetic energy due to the mean velocity gradients, Y_M accounts for the effects of compressibility³⁹, $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ are calibrated constants, G_b accounts for buoyancy, and σ_k and σ_ε are turbulent Prandtl numbers for k and ε . The original k - ε turbulence model also includes user-defined source terms for k and ε as well as a term to account for buoyancy, but these terms have been neglected for the sake of this paper. The terms accounting for the generation of turbulence kinetic energy and compressibility are defined as

$$G_k = \mu_t S^2 \tag{2.26}$$

$$Y_M = 2\rho\varepsilon M_t^2 \tag{2.27}$$

where

$$M_t = \sqrt{\frac{k}{a^2}} \tag{2.28}$$

The constants and the turbulent Prandtl numbers were calibrated such that the k - ε turbulence model matched experimental data for flows in both air and water, and ranging from homogeneous shear flows to decaying isotropic grid turbulence¹¹.

The k - ε turbulence model has proven successful for a wide range of relatively simple flows, but well-known limitation is that the formulation for turbulence dissipation is not applicable for wall-bounded adverse pressure gradient flows⁴⁰. The model is based on the Boussinesq assumption, making it incapable of modeling Reynolds stress anisotropy that is inherent in circulation control flows. Also, the model is semi-empirical, meaning that one transport equation (turbulence kinetic energy) was derived according to a strict mathematical derivation, whereas the other transport equation (turbulence dissipation rate) was derived according to physical reasoning, and hardly resembles the mathematically exact derivation¹¹. As a consequence, the model has needed ad-hoc damping functions to accurately model a range of flows. Also, the model cannot be integrated to the wall, and requires the use of empirical wall functions to model boundary layer flows. Furthermore, the k - ε turbulence model is also limited to flows in which streamline curvature is negligible. Heschl et al.^{41,42} demonstrated that the standard k - ε turbulence model under-predicted lateral spreading rates for a three dimensional free shear jet, and it is likely that the model would do the same for a circulation control jet. Finally, Bell⁴³ concluded in a series of test cases for comparing turbulence models that the k - ε turbulence model is not the model of choice in flows with separation, transition, or low Reynolds number effects.

2.2.d. k - ω

Wilcox's k - ω turbulence model³⁵ is a very popular two-equation model, with transport equations for turbulence kinetic energy and specific dissipation rate. The model has largely been modified on an ad-hoc basis since it was first proposed, to provide greater

accuracy for a range of flows. In particular, the model has been calibrated for free shear flows. Turbulence kinetic energy and specific dissipation rate are obtained according to the following transport equations.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k \quad (2.29)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega \quad (2.30)$$

In these transport equations, the G terms represent generation due to mean velocity gradients, the Γ terms represent effective diffusivity, and the Y terms represent dissipation due to turbulence. The effective diffusivities are defined as

$$\Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \quad (2.31)$$

$$\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega} \quad (2.32)$$

Just as for the k - ε turbulence model, the σ terms are turbulent Prandtl numbers for their respective transport variables (turbulence kinetic energy or specific dissipation rate).

The turbulent viscosity is computed as

$$\mu_t = \alpha^* \frac{\rho k}{\omega} \quad (2.33)$$

where α^* is a damping function intended to reduce turbulent viscosity in low Reynolds number regions, and is calculated as follows.

$$\alpha^* = \alpha_\infty^* \left(\frac{\alpha_0^* + \frac{Re_t}{R_k}}{1 + \frac{Re_t}{R_k}} \right) \quad (2.34)$$

where

$$Re_t = \frac{\rho k}{\mu \omega} \quad (2.35)$$

and R_k and α_0^* are calibrated constants.

The production of turbulence kinetic energy defined in the same way as for the k - ε turbulence model, where

$$G_k = \mu_t S^2 \quad (2.36)$$

The production of specific dissipation is determined as

$$G_\omega = \alpha \frac{\omega}{k} G_k \quad (2.37)$$

where

$$\alpha = \frac{\alpha_\infty}{\alpha^*} \left(\frac{\alpha_0 + \frac{Re_t}{R_\omega}}{1 + \frac{Re_t}{R_\omega}} \right) \quad (2.38)$$

and R_ω and α_0 are calibrated constants.

Turbulence dissipation is modeled as

$$Y_k = \rho \beta^* f_{\beta^*} k \omega \quad (2.39)$$

where

$$f_{\beta^*} = \begin{cases} 1 & \chi_k \leq 0 \\ \frac{1 + 680\chi_k^2}{1 + 400\chi_k^2} & \chi_k > 0 \end{cases} \quad (2.40)$$

$$\chi_k = \frac{1}{\omega^3} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (2.41)$$

and β^* is a function of calibrated constants and the turbulent Reynolds number.

Dissipation of specific dissipation rate is modeled as

$$Y_\omega = \rho \beta f_\beta \omega^2 \quad (2.42)$$

where

$$f_\beta = \frac{1 + 70\chi_\omega}{1 + 80\chi_\omega} \quad (2.43)$$

$$\chi_\omega = \left| \frac{\Omega_{ij} \Omega_{jk} S_{ki}}{(\beta_\infty^* \omega)^3} \right| \quad (2.44)$$

and β and β_∞^* are functions of calibrated constants. These terms make use of a compressibility function that accounts for high Mach-number effects.

Unlike the k - ϵ turbulence model, the standard k - ω turbulence model is valid to the wall, eliminating the need for wall functions; this, though, is due to the ad-hoc damping functions α and α^* . Damping functions are losing popularity in turbulence modeling due to their entirely non-physical formulation. In addition to the undesirable use of damping functions, the standard k - ω suffers from several other shortcomings. First, like any turbulence model based on the Boussinesq assumption, the model cannot capture Reynolds stress anisotropy. Second, the model is empirically based, meaning that both transport equations are based upon physical reasoning rather than strict mathematical derivation, and has been adjusted such that the results match experimental results. In other words, the k - ω turbulence model is fundamentally incorrect, but has been adjusted such that the results match certain experimental data. This works quite well for a limited range of flows, but such adjustments cannot adequately model all flows. The model has been primarily adjusted for free shear flows, and has not been properly calibrated for flows with high-speed jets bounded on one side by a shear layer, and on the other side by a wall, limiting its application for circulation control. Bell concluded in his turbulent flow

case studies that the $k-\omega$ turbulence model is more accurate for most flows than the $k-\varepsilon$ or Spalart-Allmaras turbulence models, but still suffered from the limitations inherent with its two-equation nature, as well as the limitations inherent with the Boussinesq assumption.

2.3. The v^2 - f Turbulence Models

2.3.a. Standard v^2 - f Turbulence Model

Durbin⁴⁴ developed the v^2 - f turbulence model to be used in flows in which near-wall turbulence is of significant importance, specifically flows with separation, recirculation, or heat transfer⁴³. The model solves four transport equations, those for turbulence kinetic energy, turbulence dissipation rate, velocity scale, and elliptic relaxation factor. The model is essentially an extension of the $k-\varepsilon$ turbulence model, with the computational advantage of using the eddy viscosity concept to close the transport equations (as opposed to full second moment closure,) but improves upon several known deficiencies of the $k-\varepsilon$ model. Specifically, the v^2 - f turbulence model can be integrated to a solid wall, eliminating the need for damping functions or wall functions⁴⁵. Also, the introduction of the velocity scale allows the model to correctly scale damping of turbulent transport near walls, which turbulence kinetic energy is theoretically incapable of⁴⁶. The model is based on the observation that the ratio k/ε is the correct time scale in turbulent flows, but that k is not the proper velocity scale. The velocity scale v^2 was introduced to alleviate this problem. Further, near to walls, the velocity components approach zero through viscous effects, but the inviscid blocking of the velocity scale has an effect on the flow field at significant distances from walls. This implies that non-local effects should be included in

turbulence models, and an elliptic differential equation (i.e., Poisson's equation) is necessary to capture correct blocking. The elliptic relaxation factor was introduced to account for the nonlocal effect of turbulence damping in the presence of walls. In recent years, the ν^2 - f turbulence model has proven robust and superior to other RANS methods, despite its linear eddy viscosity formulation and insensitivity to streamline curvature⁴³.

The ν^2 - f turbulence model uses similar transport equations for turbulence kinetic energy and turbulence dissipation rate as does the k - ϵ turbulence model. In addition to these transport equations, this model solves the following transport equations for the velocity scale and the elliptic relaxation. The transport equations for the ν^2 - f turbulence model are⁴⁷ as follows.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = P_k - \rho \epsilon + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \quad (2.45)$$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_j}(\rho \epsilon u_j) = \frac{C_{\epsilon 1}^* P_k - \rho C_{\epsilon 2} \epsilon}{T} + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] \quad (2.46)$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \overline{v^2}) + \frac{\partial}{\partial x_j}(\rho \overline{v^2} u_j) \\ = \rho k f - 6 \rho \overline{v^2} \frac{\epsilon}{k} + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial \overline{v^2}}{\partial x_j} \right] \end{aligned} \quad (2.47)$$

$$L^2 \frac{\partial f^2}{\partial x_j} - f = \frac{1}{T} (C_1 - 1) \left(\frac{\overline{v^2}}{k} - \frac{2}{3} \right) - C_2 \frac{P_k}{\rho k} \quad (2.48)$$

where P_k represents the production of turbulence kinetic energy due to the mean flow velocity gradients, and is modeled as:

$$P_k = 2 \mu_t S^2 \quad (2.49)$$

Away from solid walls, k/ε is a reasonable estimate for the turbulence time scale³². However, near solid walls, $k \rightarrow 0$ as $y \rightarrow 0$, but $\varepsilon > 0$; because of this, there is some point near the wall where k/ε becomes less than the Kolmogoroff scale $\sqrt{\nu/\varepsilon}$. Since the turbulence time scale cannot be less than the Kolmogoroff scale, the turbulence time scale is defined as follows.

$$T = \max \left[\frac{k}{\varepsilon}, C_T \sqrt{\frac{\mu}{\rho \varepsilon}} \right] \quad (2.50)$$

As for the turbulence length scale, the standard estimate of $(k^{3/2})/\varepsilon$ is valid away from solid walls, but this estimate becomes less than the Kolmogoroff scale $(\nu^3/\varepsilon)^{1/4}$ near walls. To avoid this problem, a lower bound is imposed on the turbulence length scale, as follows.

$$L = C_L \max \left[\frac{k^{3/2}}{\varepsilon}, C_\eta \left(\frac{\mu^3}{\rho^3 \varepsilon} \right)^{1/4} \right] \quad (2.51)$$

The turbulent viscosity is defined as:

$$\mu_t = \rho C_\mu \overline{v^2} T \quad (2.52)$$

Finally, the calibrated constants are:

$$\begin{aligned} C_\mu &= 0.2, \quad \sigma_k = 1, \quad C_1 = 1.6, \quad C_2 = 0.3, \\ C_T &= 6, \quad C_L = 0.23, \quad C_\eta = 60 \end{aligned} \quad (2.53)$$

Since there is a lack of agreement on the specific values for the calibration constants, for this work the constants have been calibrated to turbulent flat plate boundary layer results. For more information on the calibration of these constants, see Page 46.

It is important to note that this model does not use any wall functions or damping functions. Instead, the model uses the velocity scale (which is a measure of velocity fluctuation normal to streamlines) to damp turbulence transport near inhomogeneities, and the elliptic relaxation function to model non-local effects.

Kalitzin⁴⁷ applied the v^2 - f turbulence model to simple aerospace configurations, and compared the results to those produced by Menter's k - ω turbulence model and the Spalart-Allmaras turbulence model. Results were generated for a subsonic A-airfoil, a transonic RAE2822 airfoil, and a subsonic three-element trapezoidal wing-body. Given that the configurations are simple airfoils and a subsonic wing-body, the cases are considered validation cases for the v^2 - f turbulence model. In all cases, the results between the three models matched quite well, indicating that the v^2 - f turbulence model correctly predicts flows for simple aerospace configurations.

Bell⁴³ performed an in-depth comparison of turbulence models for a wide variety of flows; the turbulence models included in the comparison include, but are not limited to, Launder-Sharma, Spalart-Allmaras, standard k - ϵ , realizable k - ϵ , k - ϵ - RNG , standard k - ω , Menter's k - ω , standard v^2 - f . The test cases include a fully developed channel flow, an asymmetric planar diffuser, an axisymmetric afterbody, a low Reynolds number flow over a backstep, and an impinging jet. In every case in which the standard v^2 - f turbulence model was used, it produced similar or more accurate results than other RANS models.

2.3.b. Nonlinear v^2 - f Turbulence Model

One deficiency in the standard v^2 - f turbulence model is the use of the Boussinesq assumption to linearize the relationship between the Reynolds stresses and the mean

strain rate. Pettersson Reif⁴⁸ proposed a nonlinear constitutive relationship that could account for turbulence anisotropy, thereby improving the v^2 - f turbulence model's predictive capability for turbulent shear flows. The nonlinearization begins with the proposal by Pope⁴⁹ for an equilibrium solution of a second-moment closure for an incompressible, two-dimensional mean flow in a non-inertial frame,

$$\begin{aligned} \frac{\overline{\rho u_i'' u_j''}}{\bar{\rho}k} &= \frac{2}{3} \mathbf{I} - a_1 \tau_1 \mathbf{S} - a_1 \tau^2 (\mathbf{S} \mathbf{W}^* - \mathbf{W}^* \mathbf{S}) \\ &+ a_3 \tau^2 \left(\mathbf{S}^2 - \frac{1}{3} |\mathbf{S}|^2 \mathbf{I} \right) \end{aligned} \quad (2.54)$$

where the a_i coefficients are functions of turbulent quantities, and τ is a turbulence time scale, and $|\mathbf{S}|^2 = S_{lk} S_{kl}$ (where $\mathbf{S}$ is a second order tensor). Pettersson Reif used this form to propose the following relation for the v^2 - f turbulence model:

$$\begin{aligned} \frac{\overline{\rho u_i'' u_j''}}{\bar{\rho}k} &= \frac{2}{3} \mathbf{I} - C_{\mu 1}^* \frac{\bar{v}^2}{k} \tau_1 \mathbf{S} - C_{\mu 2}^* V_1 \tau^2 (\mathbf{S} \mathbf{W}^* - \mathbf{W}^* \mathbf{S}) \\ &+ C_{\mu 3}^* V_2 \tau^2 \left(\mathbf{S}^2 - \frac{1}{3} |\mathbf{S}|^2 \mathbf{I} \right) \end{aligned} \quad (2.55)$$

where V_i and $C_{\mu i}^*$ are functions of $\frac{\bar{v}^2}{k}$ and dimensionless velocity-gradient parameters

$$\begin{aligned} \eta_1 &= \tau^2 S_{ik} S_{ik} \\ \eta_2 &= \tau^2 W_{ik}^* W_{ik}^* \end{aligned} \quad (2.56)$$

For two-dimensional incompressible flow, Equation (2.55) gives the following nonzero components of the Reynolds stress tensor.

$$\frac{\overline{\rho (u_1'')^2}}{\bar{\rho}k} = \frac{2}{3} - 2C_{\mu 1}^* \frac{v^2}{k} \tau_1 \lambda + \frac{1}{3} C_{\mu 3}^* V_2 \tau^2 \lambda^2 \quad (2.57)$$

$$\frac{\overline{\rho(u_2'')^2}}{\bar{\rho}k} = \frac{2}{3} + 2C_{\mu 1}^* \frac{v^2}{k} \tau_1 \lambda + \frac{1}{3} C_{\mu 3}^* V_2 \tau^2 \lambda^2 \quad (2.58)$$

$$\frac{\overline{\rho(u_3'')^2}}{\bar{\rho}k} = \frac{2}{3} - \frac{1}{3} C_{\mu 3}^* V_2 \tau^2 \lambda^2 \quad (2.59)$$

$$\frac{\overline{\rho u_1'' u_2''}}{\bar{\rho}k} = -2C_{\mu 2}^* V_1 \tau^2 \omega \lambda \quad (2.60)$$

In the model proposed by Pettersson Reif, the principles of realizability and internal consistency were used to determine the final form of the nonlinear v^2 - f turbulence model. Realizability is the requirement that all quantities known to be strictly positive must be guaranteed to be positive by the turbulence model³⁵. Specifically, for the Reynolds stresses, the following three relations constitute realizability.

$$\begin{aligned} \overline{\rho u_\alpha^2} &\geq 0 \\ \overline{\rho u_\alpha^2} &\leq \bar{\rho}k \\ \bar{\rho}(\overline{u_\alpha u_\beta})^2 &\leq \overline{\rho u_\alpha^2 u_\beta^2} \end{aligned} \quad (2.61)$$

It was assumed that the coefficient $C_{\mu 1}^*$ would take the same form as that proposed by Pettersson Reif et al.⁵⁰ for the streamline curvature corrected v^2 - f turbulence model. For more information on the streamline curvature corrected v^2 - f turbulence model, see Page 40. Applying the constraints in Equation (2.61) to Equations (2.57)-(2.60) results in the following forms for the coefficients $C_{\mu i}^*$:

$$C_{\mu 1}^* = C_\mu F \quad (2.62)$$

$$C_{\mu 2}^* = \frac{\beta_0 A}{\beta_1 + V_{10} \sqrt{\eta_1 \eta_2}} \quad (2.63)$$

$$C_{\mu 3}^* = \frac{\gamma_0}{\gamma_1 + V_{20}\eta_1} \quad (2.64)$$

where $\beta_0 = \gamma_0 = 1$ and the coefficients $\beta_1 \ll 1$ and $\gamma_1 \ll 1$ exist only to prevent singularities.

The remaining unknowns can be determined by applying the concept of internal consistency, according to which the nonlinear constitutive relation should reduce to $\overline{u_2^2} \approx \overline{v^2}$ in parallel shear flow and an inertial reference frame. In such a flow, $S_{12} = S_{21} = \Omega_{12}^* = -\Omega_{21}^*$, and the diagonal elements of the Reynolds stress tensor can be written as:

$$\frac{\overline{\rho(u_1'')^2}}{\bar{\rho}k} = \frac{2}{3} + C_{\mu 2}^* V_1 \eta_1 + \frac{1}{6} C_{\mu 3}^* V_2 \eta_1 \quad (2.65)$$

$$\frac{\overline{\rho(u_2'')^2}}{\bar{\rho}k} = \frac{2}{3} - C_{\mu 2}^* V_1 \eta_1 + \frac{1}{6} C_{\mu 3}^* V_2 \eta_1 \quad (2.66)$$

$$\frac{\overline{\rho(u_3'')^2}}{\bar{\rho}k} = \frac{2}{3} - \frac{1}{3} C_{\mu 3}^* V_2 \eta_1 \quad (2.67)$$

Internal consistency requires that $\overline{u_2^2}$ reduces to $\overline{v^2}$ in parallel shear flow and an inertial reference frame; that is,

$$\frac{\overline{\rho(u_2'')^2}}{\bar{\rho}k} \rightarrow \frac{\overline{\rho v^2}}{\bar{\rho}k} = \frac{2}{3} - \beta_0 A \frac{V_1}{V_{10}} + \frac{1}{6} \gamma_0 \frac{V_2}{V_{20}} \quad (2.68)$$

Given that

$$\frac{V_1}{V_{10}} = \frac{V_2}{V_{20}} = \frac{6}{5} \left(\frac{2}{3} - \frac{\overline{v^2}}{k} \right) \quad (2.69)$$

the nonlinear v^2 - f turbulence model is fully defined. The final model is described by the following equation set.

$$\begin{aligned} \frac{\overline{\rho u_i'' u_j''}}{\bar{\rho}} = & \frac{2}{3} k \delta_{ij} - 2 C_{\mu 1}^* \bar{v}^2 \tau_1 S_{ij} \\ & - V \tau^2 \left[C_{\mu 2}^* (S_{ik} W_{kj}^* + S_{jk} W_{ki}^*) \right. \\ & \left. - C_{\mu 3}^* \left(S_{ik} S_{kj} - \frac{1}{3} |S^2| \delta_{ij} \right) \right] \end{aligned} \quad (2.70)$$

where

$$C_{\mu 1}^* = C_{\mu} F \quad (2.71)$$

$$C_{\mu 2}^* = \frac{6}{5} \frac{\sqrt{1 - \left(C_{\mu 1}^* \frac{\bar{v}^2}{k} \right)^2} 2\eta_1}{\beta_1 + \sqrt{\eta_1 \eta_2}} \quad (2.72)$$

$$C_{\mu 3}^* = \frac{6}{5} \frac{1}{\gamma_1 + \eta_1} \quad (2.73)$$

$$V = \max \left(\frac{2}{3} - \frac{\bar{v}^2}{k}, 0 \right) \quad (2.74)$$

$$\beta_1 = \frac{1}{0.1 + \sqrt{\eta_1 \eta_2}} \quad (2.75)$$

$$\gamma_1 = \frac{1}{0.1 + \eta_1} \quad (2.76)$$

The calibrated constants for the nonlinear v^2 - f turbulence model are the same as those for the linear v^2 - f turbulence model. For more information on the calibration of these constants, see Page 46.

Heschl et al.⁴² used the nonlinear formulation of the v^2 - f turbulence model to solve a flow in a room with three-dimensional wall jets, and compared the results their experimental data collected using PIV as well as results for many other turbulence models. The nonlinear v^2 - f turbulence model proved robust enough to capture secondary flows generated by the wall jets, a flow phenomenon that is attributed to Reynolds stress anisotropy and was not predicted by any other turbulence models. These results are shown below.

Figure 2-1: Comparison of turbulence models from Heschl et al.

2.3.c. Streamline Curvature Corrected Nonlinear v^2 - f Turbulence Model

Another deficiency in the v^2 - f turbulence model is its insensitivity to streamline curvature. In the standard v^2 - f turbulence model, the anisotropy tensor is

$$b_{ij} = C_\mu \frac{v^2}{k} \frac{k}{\varepsilon} S_{ij} \quad (2.77)$$

and in the nonlinear v^2 - f turbulence model, the anisotropy tensor is

$$b_{ij} = \frac{\overline{\rho u_i'' u_j''}}{\bar{\rho} k} - \frac{2}{3} \delta_{ij} \quad (2.78)$$

These formulations are based on the assumption of Galilean invariance, according to which the anisotropy tensor should be the same in any coordinate system. Unfortunately, for flows with streamline curvature, this formulation of the anisotropy tensor is only valid in a streamline-oriented coordinate system. As a consequence of this, Duraisamy⁵¹ concludes that the anisotropy tensor is unable to capture streamline curvature effects, and these effects need to be incorporated explicitly. The formulation for streamline curvature effects begins with work by Pettersson Reif et al.⁵⁰, in which the ν^2 - f turbulence model was sensitized to frame-rotation effects (a non-inertial effect that also needed to be explicitly introduced). This work consequently sensitized the vorticity invariant term to the mean vorticity tensor, which accounts for some streamline curvature effects. This work led to the following formulations for the strain and vorticity invariants:

$$\eta_1 = \frac{k^2}{\varepsilon^2} \left\| \left[\frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] \right\|^2 \quad (2.79)$$

$$\eta_2 = \frac{k^2}{\varepsilon^2} \left\| \left[\frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \right] + C_\omega \Omega_{ij} \right\|^2 \quad (2.80)$$

$$\eta_3 = \eta_1 - \eta_2 \quad (2.81)$$

where Ω_{ij} is the angular velocity of the reference frame and $|\mathbf{X}|^2 = X_{lk} X_{kl}$ (where $\mathbf{X}$ is a second order tensor). The angular velocity term is neglected in the formulation of the streamline curvature corrected ν^2 - f turbulence model, as turbulence models applied to circulation control flows do not need to account for reference frame rotation.

The turbulent viscosity coefficient is sensitized as follows.

$$C_\mu^*(\eta_1, \eta_2) = C_\mu \frac{1 + \alpha_2 |\eta_3| + \alpha_3 |\eta_3|}{1 + \alpha_4 |\eta_3|} \left[\sqrt{\frac{1 + \alpha_5 \eta_1}{1 + \alpha_5 \eta_2}} + \alpha_1 \sqrt{\eta_2} \sqrt{|\eta_3| - \eta_3} \right]^{-1} \quad (2.82)$$

where the α_i coefficients are

$$\begin{aligned} \alpha_1 &= 0.055 \sqrt{f_1} \\ \alpha_2 &= \frac{1}{2} f_1 \\ \alpha_3 &= \frac{1}{4} f_1 \\ \alpha_4 &= \frac{1}{5} \sqrt{f_1} \\ \alpha_5 &= \frac{1}{40} \end{aligned} \quad (2.83)$$

and

$$f_1 = \frac{\sqrt{\left(\frac{v^2}{k}\right)}}{\sqrt{\left(\frac{v^2}{k}\right)_\infty}}, \quad \left(\frac{v^2}{k}\right)_\infty = 0.367 \quad (2.84)$$

Note that in homogeneous shear flow, $\left(\frac{v^2}{k}\right) = \left(\frac{v^2}{k}\right)_\infty$, so $f_1 = 1$ and there is no need for, or application of, curvature correction.

Duraisamy⁵¹ has shown that this formulation of the v^2 - f turbulence model, which is sensitized for frame-rotation effects, has shown some improvement for flows with streamline curvature, as is shown in Figure 2-2. In a wingtip vortex, the standard v^2 - f turbulence model predicts high eddy viscosity at the center of the vortex; this is unphysical, as a wingtip vortex acts similar to a rotating solid body, which has a

stabilizing effect near the center of rotation. Pettersson Reif's frame-rotation modification improves prediction by reducing the eddy viscosity near the center of the vortex, but still shows relatively high eddy viscosity near the exterior of the vortex. Duraisamy et al. further modified the v^2 - f turbulence model to explicitly incorporate a streamline curvature correction.

The explicit introduction of streamline curvature correction was approached by adding an antisymmetric objective vorticity tensor that results from a transformation from the global coordinate frame to a streamline-oriented coordinate frame. Methods for determining this tensor are generally classified into two categories, acceleration-based and strain-based, of which the latter has proven more robust²⁶. For the v^2 - f turbulence model, the vorticity invariant is redefined to include this new term:

$$\eta_2 = \left(\frac{k}{\epsilon}\right)^2 \left| \left[\frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \right] + C_\omega (\Omega_{ij} + \bar{\Omega}_{ij}) \right| \quad (2.85)$$

where, for non-rotating reference frames, $\Omega_{ij} = 0$ and $\bar{\Omega}_{ij} = -\epsilon_{ijk} \bar{\omega}_k$ is the objective vorticity tensor. According to Wallin et al.²³, the vorticity term is determined as:

$$\bar{\omega}_i = \frac{\Pi_1^2 \delta_{ij} + 12\Pi_2 S_{ij} + 6\Pi_1 S_{ik} S_{kj}}{2\Pi_1^3 - 12\Pi_2^2} S_{pl} S'_{lq} \epsilon_{pqj} \quad (2.86)$$

where $\Pi_1 = \text{trace}(\mathbf{S}^2)$, $\Pi_2 = \text{trace}(\mathbf{S}^3)$, and $(\ )'$ is a material derivative. The Einstein summation notation of Equation (2.86) is convenient for its conciseness, but is not well-suited for programming. For the sakes of clarity and completeness, the expansion of the objective vorticity tensor $\bar{\Omega}_{ij}$ is shown as follows.

$$\begin{bmatrix} \bar{\Omega}_{11} & \bar{\Omega}_{12} & \bar{\Omega}_{13} \\ \bar{\Omega}_{21} & \bar{\Omega}_{22} & \bar{\Omega}_{23} \\ \bar{\Omega}_{31} & \bar{\Omega}_{32} & \bar{\Omega}_{33} \end{bmatrix} = \begin{bmatrix} 0 & -\omega_3 & \omega_2 \\ \omega_3 & 0 & -\omega_1 \\ -\omega_2 & \omega_1 & 0 \end{bmatrix} \quad (2.87)$$

where

$$\begin{aligned} \begin{bmatrix} \omega_1 \\ \omega_2 \\ \omega_3 \end{bmatrix} &= \frac{1}{2\Pi_1^3 - 12\Pi_2^2} \left(\Pi_1^2 \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \right. \\ &\quad + 12\Pi_2 \begin{bmatrix} S_{11} & S_{12} & S_{13} \\ S_{21} & S_{22} & S_{23} \\ S_{31} & S_{32} & S_{33} \end{bmatrix} \\ &\quad \left. + 6\Pi_1 \alpha \begin{bmatrix} S_{11} & S_{12} & S_{13} \\ S_{21} & S_{22} & S_{23} \\ S_{31} & S_{32} & S_{33} \end{bmatrix}^2 \right) \begin{bmatrix} S_{23}S'_{23} - S_{32}S'_{32} \\ S_{31}S'_{31} - S_{13}S'_{13} \\ S_{12}S'_{12} - S_{21}S'_{21} \end{bmatrix} \end{aligned} \quad (2.88)$$

The calibrated constants for the nonlinear v^2 - f turbulence model with streamline curvature correction are the same as those for the nonlinear v^2 - f turbulence model. For more information on the calibration of these constants, see Page 46.

The curvature corrected v^2 - f turbulence model has been applied to a NACA 0012 airfoil, and results were compared to the standard v^2 - f turbulence model, the model with frame-rotation effects, and experimental data collected by Chow et al.⁵² These results indicate considerable improvement for wingtip vortex flows, both in prediction of the mean velocity field and turbulence kinetic energy, though Duraisamy et al. warn that the results should be considered preliminary because of the lack of other validation cases. The following figures show the vertical velocity components for the models and experimental results, demonstrating the improvement of the streamline curvature correction.

(a) Baseline $v^2 - f$

(b) Modified $v^2 - f$ (no CC)

(c) Modified $v^2 - f$ (with CC)

Figure 2-2: Eddy viscosity contours (normalized by molecular viscosity) at $x/c = 0.246$ for a wingtip vortex flow, from Duraisamy et al.

(a) $x/c = -0.296$

(b) $x/c = -0.114$

(c) $x/c = 0.125$

(d) $x/c = 0.456$

Figure 2-3: Vertical velocity (normalized by free-stream velocity) along a line passing through a wingtip vortex core, from Duraisamy et al.

2.3.d. Calibration of Model Constants

The constants used in the v^2 - f turbulence model have been altered and calibrated numerous times since the model was first introduced. Many authors make logical arguments for the order of magnitude of the constants, but the constants have yet to be

systematically calibrated (at least as far as the author of this paper is aware). Further, the published values for many constants vary considerably between papers. The following table demonstrates how much some of these constants have varied between publications.

Table 2-1. Published Constants for ν^2 - f Turbulence Models

	C_μ	C_1	C_L	C_η
Durbin ⁴⁴	0.20	1.2	0.17	80
Kalitzin ⁵³ (Model 1)	0.19	1.4	0.30	70
Kalitzin ⁵³ (Model 2)	0.22	1.4	0.23	70
Laurence ⁵⁴	0.22	1.4	0.25	110
Lien ⁵⁵	0.22	1.4	0.23	70

To address this issue, the constants were calibrated for a turbulent flow over a flat plate. The constants CE_D , CE_2 , σ_k , σ_ε , C_μ , C_1 , C_2 , C_L , and C_η were parametrically varied, while particular attention was paid to two aspects of the flat plate flow. First, the skin friction profile needed to match the well-known experimental profile defined as follows⁵⁶.

$$C_f = \frac{0.027}{(Re_x)^{(1/7)}} \quad (2.89)$$

CE_D , CE_2 , σ_k , and σ_ε had very minor effects on the skin friction profiles, and were left as their previously published values. The effects of varying the remaining constants on the skin friction profile are shown in Figure 2-4.

Figure 2-4. Calibration curves: C_1 (top-left); C_2 (top-right); C_μ (middle-left); C_η (middle-right); C_L (bottom)

Second, the model was tuned such that the near-wall velocity profile matched the widely known velocity profiles in the linear sublayer, buffer region, and into the log region. The velocity profile in the linear sublayer follows a linear relationship between the velocity and distance from the wall, the velocity profile in the buffer layer is defined by Spalding's equation, and velocity profile in the logarithmic overlap region follows a logarithmic relationship¹⁷. These three relationships are, respectively,

$$\begin{aligned}
 u^+ &= y^+ & \text{for } y^+ < 5 \\
 u^+ &= -y^+ + e^{-\kappa B} \left[e^{\kappa u^+} - 1 - \kappa u^+ - \frac{1}{2}(\kappa u^+)^2 - \frac{1}{6}(\kappa u^+)^3 \right] & \text{for } 5 < y^+ < 30 \\
 u^+ &= \frac{1}{\kappa} \ln(u^+) + B & \text{for } 30 < y^+ < 350
 \end{aligned}
 \tag{2.90}$$

where $\kappa = 0.41$ and $B = 5.0$.

The boundary layer velocity profiles were monitored during the parametric calibration process. The resulting velocity profile can be found on Page 56.

This calibration process resulted in the following constants.

Table 2-2. Calibrated Constants for v^2 - f Turbulence Models

C_μ	C_1	C_2	C_L	C_η	CE_D	CE_2	σ_k	σ_ε
0.2	1.6	0.3	0.23	60	0.05	1.9	1.0	1.3

3. Use of User-Defined Functions with FLUENT

3.1. User-Defined Functions, Scalars, and Memory

A standard CFD interface cannot be written to anticipate all possible needs. To address this, the functionality of FLUENT was improved by allowing the user to define custom user-defined functions (UDFs). UDFs are routines, programmed in C, that

dynamically link with the FLUENT solver. UDFs allow the use of standard C functions, including mathematical operations and logic statements, as well as many pre-programmed FLUENT macros. Via these functions, a FLUENT user can access and customize boundary conditions, material properties, reaction rates, source terms, and diffusivity functions, and can adjust computed values on a once-per-iteration basis; all of these are necessary to define a new turbulence model. Source terms can be defined in two ways; first, a pre-existing source term (for example, turbulence kinetic energy) can be modified using a UDF. Second, a new source term can be defined as a user-defined scalar (UDS,) to be transported according to a custom transport equation of the form:

$$\frac{\partial \Phi_k}{\partial t} + \frac{\partial}{\partial x_i} \left(F_i \Phi_k - \Gamma_k \frac{\partial \Phi_k}{\partial x_i} \right) = S_{\Phi_k} \quad (3.1)$$

where k is an index associated with each UDS, F is a user-defined flux function, S is a user-defined source term, Γ is a user-defined diffusivity function, and Φ is the UDS. Finally, FLUENT allows a user to customize memory allocation via user-defined memory locations (UDMs). All values must be returned to FLUENT in SI units.

FLUENT allows UDFs to be read in two ways: as interpreted functions or as compiled functions. Reading a UDF as interpreted requires an internal interpreter to compile the code on a line-by-line basis. Reading a UDF as compiled translates the source code to machine language in one step. Interpreted UDFs are generally slower, require additional memory, and offer only a limited subset of FLUENT's macros. All of the ν^2 - f turbulence models in this paper were read as compiled UDFs.

The general architecture for user access to the FLUENT solver is shown in the following figure⁵⁷.

Figure 3-1: Architecture for user access to the FLUENT solver

Applying a UDF to a FLUENT case requires five general steps. These steps will be elaborated upon in the following section.

1. Create the UDF source code
2. Create a FLUENT case file
3. Compile the UDF
4. Attach the variables in the UDF to the FLUENT solver
5. Solve

3.2.Implementation of the ν^2 - f Turbulence Models

3.2.a. Source Code

The ν^2 - f turbulence models were written for use in FLUENT via user-defined functions. For each transported variable, FLUENT solves Equation (3.1). The turbulence

kinetic energy k , turbulence dissipation rate ϵ , the velocity scale v^2 , and the elliptic relaxation factor f were transported according to this equation, where the associated terms were defined as follows.

Table 3-1. User-defined function terms for the v^2 - f turbulence models

Transported variable, Φ	Index, k	Flux function, F	Source term, S	Diffusivity, Γ
Turbulence kinetic energy, k	0	$\dot{m}$	$\mu_t - \rho^2 C_\mu v^2 \frac{k}{P_k}$	$\frac{\mu_t}{\sigma_k} - \mu_l$
Turbulence dissipation rate, ϵ	1	$\dot{m}$	$\frac{CE_1 P_k}{T} - \frac{CE_2 \rho \epsilon}{T}$	$\frac{\mu_t}{\sigma_k} - \mu_l$
Velocity scale, v^2	2	$\dot{m}$	$\frac{\rho k f - 6 \rho v^2}{T}$	$\frac{\mu_t}{\sigma_\epsilon} - \mu_l$
Elliptic relaxation factor, f	3	0	$-\frac{f}{L^2} - \frac{C_1 - 6}{T^2 L} \left[\frac{v^2}{k} - \frac{2}{3} (C_1 - 1) \right] + \frac{C_2 P_k}{\rho k T^2}$	0

In addition to the transport equations for k , ϵ , v^2 , and f , an adjust function is necessary to modify FLUENT variables that are not passed as arguments at every iteration. In particular, an adjust function was required to define the turbulent viscosity, the turbulent time and length scales, the turbulent production, the mean strain rates and Reynolds stresses (in the case of the nonlinear v^2 - f turbulence models,) and the curvature correction terms (in the case of the nonlinear v^2 - f turbulence model with streamline curvature correction).

Finally, any boundary conditions that are not specified as a value of zero or a gradient of zero were defined using a boundary condition macro. The only boundary condition

that needed this treatment was that of the turbulence dissipation rate at physical walls, where $\epsilon_{bc} \rightarrow \frac{2\mu k}{\rho y^2}$.

For the linear and nonlinear v^2 - f turbulence models, a single source file containing the above-mentioned code is all that is needed. However, the nonlinear v^2 - f turbulence model with streamline curvature correction requires matrix operations (for the specific operations, see Page 40). These matrix operations require additional code, since matrix math cannot be performed with the programming language C. This additional code was written by Dr. David Marshall specifically for this work.

3.2.b. Coupling the Source Code to FLUENT

Because gradients are required for the v^2 - f turbulence models to work properly and the source code does not specify substitutes for those gradients at the first iteration, the process of implementing the v^2 - f turbulence models in FLUENT is more involved than the process by which one would implement one of FLUENT's native turbulence models. The general outline is as follows.

1. Load the mesh file. Set up the boundary conditions and material properties for the k - ϵ turbulence model and iterate several times.
2. Compile the user-defined function for the source code via the *Define – User-Defined – Functions – Compiled* menu. For the linear and nonlinear v^2 - f turbulence models, add the v^2 - f source code to the *Source Files* list. For the nonlinear v^2 - f - cc turbulence model, add both the v^2 - f - cc source code and the matrix library source code to the *Source Files* list, and add the matrix library header file to the *Header Files* menu. Give the UDF library a path and name,

and build the UDF. Load the UDF from the same window. It is important to note that UDFs compiled in parallel mode will not run in serial mode, and vice-versa.

3. Define the user-defined scalars via the *Define – User-Defined – Scalars* menu. For the linear and nonlinear v^2 - f turbulence models, set the number of user-defined scalars to 8; for the nonlinear v^2 - f - cc turbulence model, set the number of user-defined scalars to 17. *Solution Zones* should be set to “all fluid zones,” while *Flux Function* should be set to “user_flux”.
4. Define the user-defined memory allocation via the *Define – User-Defined – Memory* menu. Set the number of UDM locations to 4.
5. Execute the two on-demand functions via the *Define – User-Defined – Execute-on-Demand* menu. Select and execute the “on_demand_calc” and “rename_UDvars” functions in succession. The first function initializes the user-defined scalar values, while the second function renames the UDS and UDM variables.
6. Define the source terms for the k , ϵ , v^2 , and f variables via the *Define – Boundary-Conditions* menu. Select the fluid zone. Under the *Source Terms* tab, add the appropriate source term for each of the k , ϵ , v^2 , and f equations.
7. Define the boundary conditions for other boundaries via the *Define – Boundary-Conditions* menu. Select each boundary, and edit the boundary conditions via the *UDS* tab. For inlets and outlets, k and ϵ should be set as they would be set for the standard k - ϵ turbulence model. Alternatively, inlet

and outlet functions are included in the source code that define 10% turbulence intensity and a turbulent viscosity ratio of 10. At walls, the values for k , v^2 , and f should be specified as zero, and the value for ϵ should be set to the “e_bc” equation.

8. Define the UDS diffusivity via the *Define – Materials* menu. *UDS Diffusivity* should be set to the “ke_v2f_diffusivity” function.
9. Turn the UDS equations off via the *Solve – Controls – Solution* menu. At this point, only the *Flow* equations should be turned on.
10. Iterate several times to initialize the UDS gradients. For the GACC solutions, 20 iterations proved adequate.
11. Add the user-defined function hook via the *Define – User-Defined – Function Hooks* menu. Add either the “ke_adjust” or “ke_adjust_lin” function hook to the *Adjust* function hooks, as appropriate.
12. Iterate several times. For the GACC solutions, 20 iterations proved adequate.
13. Turn on the UDS equations one at a time via the *Solve – Controls – Solution* menu, iterating several times before new equations are activated. For the GACC solutions, 500 iterations per equation proved adequate.
14. Select the standard k - ϵ turbulence model with enhanced wall treatment via the *Define – Models – Viscous* menu. Change the *Turbulent Viscosity* function to “user_mu_t”.
15. Iterate to convergence.

As for standard turbulence models, stability is enhanced if first order solutions are obtained before the discretization schemes are defined as second order. Under-relaxation factors may need to be lowered if the solution diverges.

4. Validation Cases

4.1. Flat Plate in Turbulent, Subsonic Flow

The three v^2 - f turbulence models and several common turbulence models were again used to solve a subsonic flow over the same flat plate, but the fluid viscosity was reduced to quicken the transition to turbulent flow. The Reynolds number at the end of the plate is 1×10^7 , indicating nearly fully turbulent flow (transition to turbulence occurs at 5% of the chord). The skin friction coefficient generated using the v^2 - f turbulence models, several common turbulence models, and a theoretical relationship are shown in the following figure. The theoretical relationship is the well-known relationship for turbulent flat plate skin friction coefficient⁵⁶,

$$C_f = \frac{0.027}{(Re_x)^{(1/7)}} \quad (4.1)$$

Figure 4-1: Skin friction coefficient distribution for a turbulent flat plate in subsonic flow

The three v^2-f turbulence models predict nearly identical skin friction coefficient distributions, again indicating that the models reduce to the base model in simple flows. Further, the models match Prandtl's skin friction coefficient distribution nearly exactly.

The boundary layer velocity profiles for the turbulent flat plate are shown in the following figure. Clearly, these results match both the expected relationships presented above, as well as experimental data generated by Wieghardt⁵⁸.

Figure 4-2: v^2 - f boundary layer velocity profiles for a turbulent flat plate

These results are consistent with those produced by standard turbulence models, as is shown in the following figure.

Figure 4-3: Boundary layer velocity profiles for a turbulent flat plate

4.2.S3H4 2D Hill

The flow over a 2D sinusoidal hill was used as the final validation case. The hill geometry definition is consistent with that defined by Kim et al.⁵⁹, where $SxHy$ denotes a maximum slope of 0.x and a height of y. The hill geometry is defined as follows.

$$y = \frac{H}{2} \left\{ 1 + \cos \left[\left(\frac{\pi}{2} \right) \left(\frac{x}{L_1} \right) \right] \right\} \quad (4.2)$$

where L_1 is the half-length of the hill, defined as

$$L = \frac{H}{2S} \quad (4.3)$$

Velocity profiles generated with the ν^2 - f turbulence models were compared to profiles generated with standard turbulence models and experimental data gathered by Kim et al. The streamline curvature and potential for separation makes the case slightly more complex than the previous validation cases, with the potential to demonstrate an improvement in the nonlinear eddy viscosity or sensitization to streamline curvature. All boundary conditions, including the logarithmic velocity profile for the inlet, were defined following Pirkl et al.⁶⁰

Experimental data shows no separation region downstream of the hill, while the Spalart-Allmaras and linear ν^2 - f turbulence models show separation. The k- ϵ , k- ω , and nonlinear ν^2 - f turbulence models show no separation, consistent with experimental data. The reason for the separation region in the linear ν^2 - f results is unknown, but is apparently solved by the inclusion of a nonlinear eddy viscosity formulation. The nonlinear ν^2 - f results and the nonlinear ν^2 - f with streamline curvature correction results are similar, indicating that streamline curvature is not a significant component of this flow. The velocity profiles are shown in the following figures.

Figure 4-4: S3H4 2D hill velocity profiles (solid: experimental; line: CFD).

5. Circulation Control Airfoil Results (2D)

5.1. Previous CFD Results

As has been repeatedly demonstrated, common turbulence models are ill-suited for circulation control flows. This can be due to inappropriate simplification of flows (in the case of zero-, one-, or two-equation eddy viscosity models) or due to tight coupling of Reynolds stress transport equations (in the case of Reynolds stress transport models). In general, CFD solutions using standard eddy viscosity models showed a significant over-prediction of integrated quantities, including the lift coefficient. For example, see results from Lee-Rausch et al. (Figure 1-4), Jones et al. (Figure 1-5), and Swanson et al. (Figure 1-6).

5.2. CFD Results with the ν^2 - f Turbulence Models

The ν^2 - f turbulence models were applied to the General Aviation Circulation Control airfoil (the same airfoil used in the studies by Jones et al. and Lee-Rausch et al.) Because of the increased robustness of the ν^2 - f turbulence models, it was expected that these models should yield increased accuracy in complex circulation control flows. Further, results were generated using more common turbulence models to verify the results presented by Lee-Rausch et al. and Jones et al.

Figure 5-1: GACC airfoil

For all cases, boundary conditions were defined to match experimental cases. The boundary conditions are as follows.

Table 5-1. Boundary conditions for GACC airfoil CFD cases

	Boundary Condition										
	$P \text{ (Pa)}$	$\rho \left(\frac{kg}{m^3} \right)$	$T \text{ (K)}$	$a \left(\frac{m}{s} \right)$	$M \text{ (-)}$	$k \left(\frac{kg}{ms^3} \right)$	$\epsilon \left(\frac{kg}{ms^4} \right)$	$v^2 \left(\frac{m}{s} \right)$	$f \text{ (-)}$	$R_{ij} \text{ (-)}$	$S_{ij} \text{ (-)}$
Wall	-	-	-	-	-	$k = 0$	$\epsilon \rightarrow \frac{2\mu k}{\rho y^2}$	$v^2 = 0$	$\frac{\delta f}{\delta y} = 0$	$R_{ij} = 0$	$S_{ij} = 0$
Pressure Farfield	101325	1.22	288	340	0.084	$k = 0.15(Ma)^2$	$\epsilon = \frac{0.09k^2}{10\nu}$	$v^2 = \frac{2}{3}k$	$\frac{\delta f}{\delta y} = 0$	$\frac{\delta R_{ij}}{\delta y} = 0$	$\frac{\delta S_{ij}}{\delta y} = 0$
Mass Flow Inlet (CC Slot)	101325	1.22	288	340	Varies	$k = 0.15(Ma)^2$	$\epsilon = \frac{0.09k^2}{10\nu}$	$v^2 = \frac{2}{3}k$	$\frac{\delta f}{\delta y} = 0$	$\frac{\delta R_{ij}}{\delta y} = 0$	$\frac{\delta S_{ij}}{\delta y} = 0$

Careful attention was paid to the grid generation process to limit errors due to poor gridding. In particular, several aspects of the grid were scrutinized. First, gridline orthogonality needed to be enforced to minimize numerical error. This criterion, along with the large flap deflection, led to a unique farfield configuration; this was necessary to provide adequate mapping of gridlines from both the flap and the near-flap region on the lower surface to the farfield. Second, since the v^2 - f turbulence model does not use damping functions nor wall functions, the cell nearest any wall needed to be placed in the laminar sublayer (generally $y^+ < 5$). Third, the leading edge discretization needed to be sufficient to capture the stagnation point, which is crucial in predicting the lift coefficient. Finally, the grid needed sufficient resolution in the wake region to capture recirculation, should the v^2 - f turbulence model predict it. A fully structured grid was generated to meet these criteria. The computational grid is shown in the following figures.

Figure 5-2: Computational grid for GACC airfoil: farfield (top-left); nearfield (top-right); leading edge (middle-left); flap (middle-right); circulation control slot (bottom)

The linear and nonlinear ν^2 - f turbulence models show improvement in the prediction of the lift coefficient for the GACC airfoil, and the results obtained using common turbulence models showed the same over-prediction that Jones et al. and Lee-Rausch et al. showed. The nonlinear ν^2 - f turbulence model with streamline curvature correction shows significant under-prediction of the lift coefficient. The reason for this is unknown, but the source is clearly in the application of streamline curvature correction. Results are shown below.

Figure 5-3: CFD results for the v^2 -f turbulence model compared to common turbulence models and experimental data.

Pressure coefficient distributions generated by standard turbulence models and the v^2 -f turbulence models for the GACC airfoil at a blowing coefficient of 0.084, and were

compared to those presented by Lee-Rausch et al. Results for the linear and nonlinear ν^2 - f turbulence models indicate reasonable agreement for the pressure coefficient for most of the airfoil, including the region surrounding the circulation control slot. However, these turbulence models produced significantly lower pressure coefficient than the experimental results and other turbulence models at the suction peak on the upper surface of the leading edge. The nonlinear ν^2 - f turbulence model with streamline curvature correction produced a lower pressure coefficient on the upper surface across the airfoil, a significantly weaker suction peak, and pressure spike near the stagnation point.

The ν^2 - f turbulence models predict the lift coefficient much more accurately than common turbulence models; however, they do not predict the pressure coefficient nearly as accurately. The under-prediction of the pressure coefficient near the suction peak contributes primarily to pressure drag rather than lift. Also, the curvature corrected ν^2 - f turbulence model shows an unusual pressure peak at the stagnation point and a weaker suction peak than the other ν^2 - f turbulence models. Because this model performed similar for simpler flows, it is likely that the curvature correction terms become significant in circulation control flows. These curvature correction terms have not been extensively validated, and may not be proper for circulation control flows.

Figure 5-4: Pressure coefficient distribution comparison for the GACC airfoil at $C_{\mu}=0.084$

Further, the v^2-f , $k-\epsilon$, and $k-\omega$ turbulence models predict a lower jet velocity than experimental results show, while the Spalart-Allmaras turbulence model predicts significantly higher jet velocity. From Equation (1.1), it can be concluded that the v^2-f , $k-\epsilon$, and $k-\omega$ turbulence models predict a higher mass flow rate and a lower jet velocity for a given blowing coefficient, while the Spalart-Allmaras turbulence model predicts a lower mass flow rate and a higher jet velocity.

Another important component of the circulation control flow field is the stagnation point. Experimental results from Jones et al. show a stagnation point at 3.01% of the

chord, while the linear ν^2-f , nonlinear ν^2-f , and curvature corrected nonlinear ν^2-f turbulence models predict the stagnation point at 1.15%, 1.27%, and 0.08%, respectively. The source of this discrepancy is unknown, but it is clear that the application of streamline curvature correction causes significant error in the circulation control cases.

Figure 5-5: Experimental stagnation point and jet velocity contours for the GACC airfoil at $C_\mu=0.084$ (Jones et al.)

Figure 5-6: Stagnation point and jet velocity comparison for the GACC airfoil at $C_{\mu}=0.084$. Top-left: Spalart-Allmaras; top-right: $k-\epsilon$; middle-left: $k-\omega$; middle-right: linear v^2-f ; bottom-left: nonlinear v^2-f ; bottom-right: nonlinear v^2-f -cc

In the wake region, two aspects of the flow are of particular note. First, the experimental data shows a small separation region aft of the flap. Contrary to this, most CFD solutions show a relatively large separation region, with the exception being the Spalart-Allmaras turbulence model, which shows a separation region similar in size to the experimental data. Second, the experimental data shows that streamlines released from the jet slot flow exit the following figures at $x/c \approx 1.30$ and $y/c \approx -0.35$. The linear and nonlinear v^2 - f turbulence models predict this jet flow relatively well, while the curvature corrected v^2 - f turbulence model shows considerable deflection in the streamlines.

Figure 5-7: Experimental jet velocity contours for the GACC airfoil at $C_\mu=0.084$ (Jones et al.)

Figure 5-8: Jet velocity comparison for the GACC airfoil at $C_{\mu}=0.084$. Top-left: Spalart-Allmaras; top-right: $k-\epsilon$; middle-left: $k-\omega$; middle-right: linear v^2-f ; bottom-left: nonlinear v^2-f ; bottom-right: nonlinear v^2-f-cc

The turbulence models used in this comparison predict similar skin friction profiles for the GACC airfoil at a blowing coefficient of 0.084, with the two exceptions. First, the Spalart-Allmaras turbulence model predicts significantly higher skin friction on the upper surface of the flap. Second, the nonlinear v^2 - f turbulence model with streamline curvature correction shows a higher skin friction coefficient on the lower surface of the airfoil, as well as a sharp drop in skin friction coefficient near the leading edge. Experimental results for the skin friction profile are not available for the GACC airfoil, and the accuracy of any of the turbulence models with regard to skin friction profiles cannot be assessed.

Figure 5-9: Skin friction coefficient distribution comparison for CFD results the GACC airfoil at $C_{\mu}=0.084$

The drag coefficients predicted by the ν^2 - f turbulence models are roughly three times higher than those predicted by standard turbulence models. The similarity between the skin friction profiles for the ν^2 - f turbulence models and the standard turbulence models indicates that the difference between the drag coefficients is primarily pressure drag.

However, experimental data for the drag coefficient is not available, so the accuracy of any of the turbulence models cannot be assessed with regard to drag.

Figure 5-10: Drag coefficient as a function of blowing coefficient for the GACC airfoil

6. Conclusion

While the ν^2 - f turbulence models show the theoretical potential to improve turbulence modeling for circulation control applications, the models appear to be incorrectly modeling some underlying physics of circulation control flows and are not producing physically sound results. The predictions for lift coefficient are considerably better for the linear and nonlinear ν^2 - f turbulence models than standard turbulence models, but this improvement is coupled with slight depreciation in the prediction of the leading edge stagnation point and pressure coefficient distributions. This indicates that some improvement can still be made in the field of turbulence modeling to improve predictions for circulation control applications.

Circulation control flow fields are complex and difficult to model. While the ν^2 - f turbulence models show potential in improving turbulence modeling for these flow fields because of the exclusion of damping functions, the inclusion of non-local effects, the nonlinearization of the eddy viscosity, and streamline curvature correction, these models have not fully solved the problem of turbulence modeling for circulation control. Further research is clearly in order in the field of turbulence modeling to solve the modeling problem for circulation control flows.

7. Future Work

This paper presents an effort to improve turbulence modeling for circulation control flows. This task has proven to be too large for a single paper, and there is still room for continuation of this work. In particular, the constants for the ν^2 - f turbulence models have been calibrated for flow over a turbulent flat plate, and this calibration process may not

be valid for circulation control flows. More calibration cases, which could include the S3H4 hill, a simple airfoil, or a curved channel flow (specifically, the curved channel used to validate the SARC model³⁶ or the curved channel used in the 1979 Stanford Olympics⁶¹), would offer significant insight into the effect of the constants on circulation control flows. Additionally, the effect of these constants on the ability of the ν^2 - f turbulence models to capture the leading edge suction peak of a circulation control airfoil would be beneficial. Finally, the ν^2 - f turbulence model codes have not been optimized, and would benefit greatly from an effort to improve their stability and rate of convergence.

REFERENCES

- ¹ Englar, R. J., and Huson, G. G., "Development of Advanced Circulation Control Wing High Lift Airfoils," AIAA Paper 83-1847, 1983.
- ² Panitz, T., and Wasan, D. T., "Flow Attachment to Solid Surfaces: the Coanda Effect," AICHE Journal vol. 18 no. 1, 1972.
- ³ Golden, R., "Design and Performance of Circulation Control Dual Radius Flap Systems," Aerospace Engineering Department, California Polytechnic State University (unpublished).
- ⁴ Jones, G. S., Yao, C. S., and Allan, B. G., "Experimental Investigation of a 2D Supercritical Circulation-Control Airfoil Using Particle Image Velocimetry," AIAA Paper 2006-3009, 2006.
- ⁵ Tritton, D. J., *Physical Fluid Dynamics*, Van Nostrand Reinhold, 1977.
- ⁶ Jingchang, L., Mao, S., and Liyi, W., "Navier-Stokes Analysis of a Circulation Control Airfoil," ACTA Mechanica Sinica, vol. 11, no. 2, 1995.
- ⁷ Liu, Y., "Numerical Simulations of the Aerodynamic Characteristics of Circulation Control Wing Sections," Georgia Institute of Technology, Atlanta, Georgia, 2003.
- ⁸ Liu, Y., Sankar, L. N., Englar, R. G., and Ahuja, K. K., "Numerical Simulations of the Steady and Unsteady Characteristics of a Circulation Control Wing Airfoil," AIAA Paper 2001-0704, 2001.
- ⁹ McGowan, G., and Gopalarathnam, A., "CFD Analysis of a Circulation Control Airfoil Using FLUENT," Mechanical and Aerospace Engineering Dept., North Carolina State Univ., Raleigh, North Carolina, 2004.
- ¹⁰ Ball, T., "CFD as Applied to the Design of Short Takeoff and Landing Vehicles Using Circulation Control," Aerospace Engineering Department, California Polytechnic State University, 2008.
- ¹¹ Lee-Rausch, E. M., Vatsa, V. N., and Rumsey, C. L., "Computational Analysis of Dual Radius Circulation Control Airfoils," AIAA Paper 2006-3012, 2006.
- ¹² Swanson, R. C., Rumsey, C. L., and Anders, S. G., "Progress Towards Computational Method for Circulation Control Airfoils," AIAA Paper 2005-29, 2005.
- ¹³ Chang, P. A., Slomski, J., Marino, T., and Ebert, M. P., "Numerical Simulation of Two- and Three-Dimensional Circulation Control Problems," 43rd AIAA Aerospace Sciences Meeting and Exhibit, 2005.
- ¹⁴ McGowan, G., Gopalarathnam, A., and Jones, S., "Analytical and Computational Study of Adaptive Circulation Control Airfoils," AIAA Paper 2004-4721, 2004.
- ¹⁵ Wilcox, D. C., "Reassessment of the Scale-Determining Equation for Advanced Turbulence Models," AIAA Journal vol. 26, pp. 1299-1310, 1988.
- ¹⁶ Launder, B. E., and Sandham, N. D., *Closure Strategies for Turbulent and Transitional Flows*, Cambridge University Press, New York, 2002.
- ¹⁷ White, F. M., *Viscous Fluid Flow*, McGraw-Hill, New York, 2006.
- ¹⁸ Pfingsten, K.-C., Jensch, C., Körber, K. W., and Radespiel, R., "Numerical Simulation of the Flow Around Circulation Control Airfoils," First CEAS European Air and Space Conference, 2007.
- ¹⁹ Novak, C. J., Cornelius, K. C., and Roads, R. K., "Experimental Investigations of the Circular Wall Jet on a Circulation Control Airfoil," AIAA Paper 1987-0155, 1987.
- ²⁰ Jones, G. S., Lin, J. C., Allan, B. G., Milholen, W. E., Rumsey, C. L., and Swanson, R. C., "Overview of CFD Validation Experiments for Circulation Control Applications at NASA," NASA 20080031119, 2008.
- ²¹ McGowan, G., "Computational Study of Adaptive Circulation Control Airfoils," Master of Science Thesis, Mechanical and Aerospace Engineering Dept., North Carolina State Univ., Raleigh, North Carolina, 2004.
- ²² Slomski, J. F., Chang, P. A., and Arunajatesan, S., "Large Eddy Simulation of a Circulation Control Airfoil," AIAA Paper 2006-3011, 2006.
- ²³ Wallin, S., and Johansson, A., "Modeling Streamline Curvature Effects in Explicit Algebraic Reynolds Stress Models," International Journal of Fluid Flow vol. 23, 2002.
- ²⁴ Spalart, P. R., and Shur, M., "On the Sensitization of Turbulence Models to Rotation and Curvature," Aerospace Science and Technology, vol. 4, 1997.
- ²⁵ Gatski, T., and Jongen, T., "Nonlinear Eddy Viscosity and Algebraic Stress Models for Solving Complex Turbulent Flows," Prog. Aero. Sci., vol. 36, 2000.
- ²⁶ Hellsten, A., Wallin, S., and Laine, S., "Scrutinizing Curvature Corrections for Algebraic Reynolds Stress Models," AIAA Paper 2002-2963, 2002.
- ²⁷ Craft, T. J., "Nonlinear Eddy-Viscosity Models," Department of Mechanical, Aerospace, and Manufacturing Engineering, University of Manchester, Manchester, England (unpublished).

- ²⁸ Smith, A. M. O., and Cebici, T., "Numerical Solution of the Turbulent Boundary Layer Equations," Douglas Aircraft Division Report DAC 33735, 1967.
- ²⁹ Baldwin, B. S., and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, 1978.
- ³⁰ Spalart, P. R., and Allmaras, S. R., "A One-Equation Turbulence Model for Aerodynamic Flows," AIAA Paper 1992-0439, 1992.
- ³¹ Harlow, F. H., and Nakayama, P. I., "Transport of Turbulence Energy Decay Rate," Los Alamos Sci. Lab., LA-3854, 1968.
- ³² Durbin, P. A., "Near-Wall Turbulence Closure Without 'Damping Functions'," Theoretical and Computational Fluid Dynamics 3:1-13, 1991.
- ³³ Pope, S. B., "A More General Effective-Viscosity Hypothesis," Journal of Fluid Mechanics, vol. 72, part 2, 1975.
- ³⁴ Edwards, J. R., and Chandra, S., "Density Corrections for Turbulence Models," Aerospace Science and Technology, Vol. 4, 2000.
- ³⁵ Wilcox, D. C., *Turbulence Modeling for CFD*, 3rd ed., DCW Industries, California, 2006.
- ³⁶ Shur, M. L., Strelets, M. K., Travin, A. K., Spalart, P. R., "Turbulence Modeling in Rotating and Curved Channels: Assessing the Spalart-Shur Correction," AIAA Journal Vol. 38, No. 5, 2000.
- ³⁷ Launder, B. E., Spalding, D. B., *Lectures in Mathematical Models of Turbulence*, Academic Press, London, England, 1972.
- ³⁸ Launder, B. E., and Sharma, B. I., "Application of the Energy-Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disk," Letters in Heat and Mass Transfer vol. 1, pp. 131-138, 1974.
- ³⁹ Sarkar, S. and Balakrishnan, L., "Application of a Reynolds-Stress Turbulence Model to the Compressible Shear Layer," ICASE Report 90-18, NASA CR 182002, 1990.
- ⁴⁰ Rumsey, C. L., Gatski, T. B., "Summary of EASM Turbulence Models in CFL3D with Validation Test Cases," NASA/TM-2003-212431, 2003.
- ⁴¹ Heschl, Ch., Sanz, W., and Klanatsky, P., "Implementation and Comparison of Different Turbulence Models for Three Dimensional Wall Jets with FLUENT," TU Graz, Institute for Thermal Turbomachinery and Machine Dynamics, 2005.
- ⁴² Heschl, C., Sanz, W., Klanatsky, P., and Madou, F., "Comparison of Different Turbulence Models to Compute Wall Affected Room Airflows," TU Graz, Institute for Turbomachinery and Machine Dynamics, 2005.
- ⁴³ Bell, B., "Turbulent Flow Case Studies," Fluent Services Center, 2003.
- ⁴⁴ Durbin, P. "Application of a Near-Wall Turbulence Model to Boundary Layers and Heat Transfer," International Journal of Heat and Fluid Flow, vol. 14, 1993.
- ⁴⁵ Parneix, S., Durbin, P. A., and Behnia, M., "Computation of 3-D Turbulent Boundary Layers Using the V2F Model," Flow, Turbulence, and Combustion, vol. 60, 1998.
- ⁴⁶ Behnia, M., Parneix, S., and Durbin, P., "Prediction of Heat Transfer in a Jet Impinging on a Flat Plate," International Journal of Heat and Mass Transfer, vol. 41, 1998.
- ⁴⁷ Kalitzin, G., "Application of the ν^2 - f Model to Aerospace Configurations," Center for Turbulence Research Annual Research Briefs, 1999.
- ⁴⁸ Pettersson Reif, B. A., "A Nonlinear Constitutive Relationship for the ν^2 - f Model," Center for Turbulence Research Annual Research Briefs, 1999.
- ⁴⁹ Pope, S. B., "A More General Eddy-Viscosity Hypothesis," Journal of Fluid Mechanics, vol. 72, 1975.
- ⁵⁰ Pettersson Reif, B. A., Durbin, P. A., and Ooi, A., "Modeling Rotational Effects in Eddy-Viscosity Closures," International Journal of Heat and Fluid Flow, 1999.
- ⁵¹ Duraisamy, K., and Iaccarino, G., "Curvature Correction and Application of the ν^2 - f Turbulence Model to Tip Vortex Flows," Center for Turbulence Research Annual Research Briefs, 2005.
- ⁵² Chow, J. S., Zilliac, G. G., and Bradshaw, P., "Mean and Turbulence Measurements in the Near Field of a Wingtip Vortex," AIAA Journal 35, 1997.
- ⁵³ Kalitzin, G., and Medic, G., "Adaptive Wall Functions for ν^2 - f Model," International Journal for Numerical Methods in Fluids, vol. 53, pp. 651-667, 2006.
- ⁵⁴ Laurence, D. R., Uribe, J. C., and Utyuzhnikov, S. V., "A Robust Formulation of the ν^2 - f Model," Flow, Turbulence, and Combustion, vol. 73, pp. 169-185, 2004.
- ⁵⁵ Lien, F., and Kalitzin, G., "Computations of Transonic Flow with the ν^2 - f Turbulence Model," International Journal of Heat and Fluid Flow, 2000.
- ⁵⁶ Kundu, P. K., *Fluid Mechanics*, Academic Press, London, 1990.
- ⁵⁷ ANSYS® FLUENT 6.2 User Guide, ANSYS, Inc.
- ⁵⁸ Wieghardt, K., and Tillmann, W., "On the Turbulent Friction Layer for Rising Pressure," NACA TM 1314, 1951.

- ⁵⁹ Kim, H. G., Lee, C. M., Lim, H. C., and Kyong, N. H., "An Experimental and Numerical Study on the Flow Over Two-Dimensional Hills," *Journal of Wind Engineering and Industrial Aerodynamics* 66, 1997.
- ⁶⁰ Pirkel, L., and Bodnar, T., "Numerical Simulation of Turbulent Flows Around Sinusoidal Hills," *Colloquium Fluid Dynamics*, 2007.
- ⁶¹ Honami, S., and Simon, T. W., "Boundary Layer Flows with Streamwise Curvature," 1979.