SIMULATION OF ENTROPY GENERATION IN LAMINAR AND BYPASS TRANSITIONAL BOUNDARY LAYER FLOWS

A Thesis

Presented in Partial Fulfillment of the Requirements for the

Degree of Master of Science

With a Major in Mechanical Engineering

In the

College of Graduate Studies

University of Idaho

by

Landon D. Owen

July 2013

Major Professor: Tao Xing, Ph.D.
Authorization to Submit Thesis

This thesis of Landon Owen, submitted for the degree of Master of Science in Mechanical Engineering and titled “Simulation of Entropy Generation in Laminar and Bypass Transitional Boundary Layer Flows,” has been reviewed in final form. Permission, as indicated by the signatures and dates given below, is now granted to submit the final copies to the College of Graduates Studies for approval.

Major Professor: 

_________________________ Date: ____________
Tao Xing

Committee Members:

_________________________ Date: ____________
Ralph S. Budwig

_________________________ Date: ____________
Chyr Pyng Liou

Department Administrator:

_________________________ Date: ____________
John C. Crepeau

Discipline’s College Dean:

_________________________ Date: ____________
Larry Stauffer

Final Approval and Acceptance by the College of Graduate Studies:

_________________________ Date: ____________
Jie Chen
Abstract

Minimizing entropy generation is important to improving the efficiency of any system. The objective of this study is to evaluate the use of computational fluid dynamics (CFD) to accurately predict entropy generation rates in laminar and bypass transitional boundary layers. The commercial CFD software, ANSYS FLUENT, is employed. Entropy generation in laminar boundary layer flows is evaluated with both favorable and adverse pressure gradients. These pressure gradients are generated using various curved slip top walls. Bypass transition is simulated using the mean inlet velocity and Reynolds stresses from the direct numerical simulation (DNS) conducted by Nolan and Zaki [1]. Various turbulence and transitional models are employed and the results are compared to the DNS data. A solution verification study is conducted on three systematically refined meshes. The factor of safety method is used to evaluate the numerical error and grid uncertainties. Monotonic convergence is achieved for the simulations in which a solution verification study is performed with the grid uncertainties less than 1.1%. The boundary layer correlation function, \( F(\lambda) \), the wall shear stress correlation function, \( S(\lambda) \), and the dissipation coefficient, \( C_d \), are calculated for the laminar CFD results. The laminar CFD results show better agreement with the correlation developed by McEligot and Walsh [2] than with the Thwaites [3, 4] correlation for \( F(\lambda) \) and \( S(\lambda) \). Overall, the percent difference between the CFD results and the correlations increase as the magnitude of the pressure gradient variable, \( \beta \), increases. The Reynolds number based on momentum thickness, \( Re_\theta \), the dissipation coefficient, \( C_d \), and the intermittency, \( \gamma \), are calculated for the bypass transition CFD results. The solvers and Reynolds averaged Navier-Stokes (RANS) turbulence models in the
transitional simulations are similar to the study by Ghasemi et al. [5], but with more accurate
inlet boundary conditions and a much finer grid. All RANS models show improvement over
Ghasemi et al. results for prediction of transition onset. The transition SST $k$-$\omega$ model results
show an accurate prediction of transition onset when compared to the DNS data. The other
RANS models show early onset of transition and higher boundary layer entropy generation
than the DNS results. Preliminary three-dimensional, unsteady simulations of bypass
transition are conducted using the large eddy simulation (LES) model with the Smagorinsky-
Lilly dynamic sub-grid-scale (SGS) model and the improved delayed detached eddy
simulation (IDDES) model. The Smagorinsky-Lilly dynamic model does not predict the
onset of transition at all. The lack of transition onset results in the under prediction of the
integral entropy generation rate throughout the transition region of the domain. The IDDES
model shows initial transition onset similar to the SST $k$-$\omega$ model but becomes fully turbulent
further downstream than predicted by the transition SST $k$-$\omega$ model.
Nomenclature

\( \langle \_ \rangle \) = Profile averaged quantity

\( \{ \_ \} \) = Function

\( \| \_ \|_2 \) = L2 norm

\( | \_ | \) = Absolute value

\( \overline{\_} \) = Time averaged quantity

\( A(x) \) = Cross-sectional area, m^2

\( h(x) \) = Height in plate-normal direction, m

\( k \) = Turbulent kinetic energy, \( \frac{u'^2 + v'^2 + w'^2}{2} \), m^2/s^2

\( L_x, L_y, L_z \) = Length in the streamwise, plate-normal, and span-wise directions, respectively, m

\( n_x, n_y, n_z \) = Number of grid points in the streamwise, plate-normal, and span-wise directions, respectively

\( P \) = Distance metric to the asymptotic range

\( p \) = Pressure, kg/(m s^2)

\( Q \) = Volumetric flow rate, m^3/s

\( q^2 \) = Sum of velocity fluctuations squared, \( u'^2 + v'^2 + w'^2 \), m^2/s^2

\( S_{ij} \) = Strain tensor, \( \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \)
\( S_T = \) Strain rate magnitude, \( \sqrt{2S_xS_y} \)

\( S_g = \) Value of a given variable with the grid refinement specified in the subscript

\( S^*, S'' = \) Entropy generation rate per unit surface area and pointwise entropy generation rate, respectively, \( \text{kg/(K s^3)} \) and \( \text{kg/(K m s^3)} \)

\( T = \) Temperature, K

\( U, u = \) Mean velocity and velocity component in the \( x \) direction, respectively, m/s

\( u_\tau = \) Friction velocity, \( \sqrt{\tau_u / \rho} \), m/s

\( u', v', w' = \) Velocity fluctuations in the \( x, y, \) and \( z \) directions, respectively, m/s

\( \overline{u'u'} = \) Mean fluctuation product in Reynolds stress, \( \text{m}^2/\text{s}^2 \)

\( V, v = \) Mean velocity and velocity component in the \( y \) direction, respectively, m/s

\( w = \) Velocity component in the \( z \) direction, m/s

\( x, y, z = \) Coordinates in the streamwise, plate-normal, and span-wise directions, respectively

**Non-Dimensional Quantities**

\( a, b = \) Constants in Thwaites and McEligot correlations

\( C_d = \) Dissipation coefficient, \( \frac{TS^*}{\rho U_{fs}^3(x)} \)

\( C_f = \) Skin friction coefficient, \( \frac{\tau_w}{\rho U_{fs}^2(x)} \)

\( C_s = \) Correlation constant

\( E_s = \) Correlation exponent

\( F[\lambda] = \) Boundary layer correlation function, \( 2(0.5C_fRe_\theta - \lambda(2 + H[\lambda])) \)
\(f, f', f''\) = Falkner-Skan function, first and second derivative, respectively

\(H(\lambda)\) = Shape factor correlation function, \(\frac{\delta^*}{\theta}\)

\(K\) = Falkner-Skan constant

\(m\) = Falkner-Skan power law parameter

\(\langle p_G \rangle\) = Profile averaged order of accuracy

\(p_{th}\) = Theoretical order of accuracy

\(Re_{(\_)}\) = Reynolds number with corresponding subscript, \(\frac{(_\_)}{\nu} U_{fs}(x)\)

\(\langle R_G \rangle\) = Profile averaged convergence ratio

\(r_G\) = Grid refinement ratio, \(\Delta x_{G_2} / \Delta x_{G_1}\) and \(\Delta x_{G_3} / \Delta x_{G_2}\)

\(S(\lambda)\) = Wall shear stress correlation function, \(\frac{\tau_{w}\theta}{\mu U_{fs}(x)}\)

\((S^*)^+\) = Entropy generation rate per unit surface area, \(\frac{TS^*}{\rho u_r^3}\)

\((S^*)^+\) = Pointwise volumetric entropy generation rate, \(\frac{T\nu S^*}{\rho u_r^4}\)

\(U_G\) = Grid uncertainty

\(U^+\) = Mean velocity, \(\frac{U}{u_r}\)

\(x^+\) = Streamwise coordinate, \(\frac{xu_r}{v}\)

\(y^+\) = Wall-normal coordinate, \(\frac{yu_r}{v}\)
Greek Symbols

$\beta = \text{Streamwise pressure gradient parameter, } \frac{2m}{1+m}$

$\Delta x, \Delta y, \Delta z = \text{Grid sizes in the x, y, and z directions, respectively, m}$

$\Delta x^+, \Delta y^+, \Delta z^+ = \text{Non-dimensional grid sizes in the x, y, and z directions, respectively}$

$\Delta t = \text{Time step size, s}$

$\Delta t^+ = \text{Non-dimensional step size, } \frac{U_\infty \Delta t}{\delta_0}$

$\delta = \text{Boundary layer thickness, m}$

$\delta^* = \text{Displacement thickness, } \int_0^\infty \left(1 - \frac{u}{U_\infty}\right) dy, \text{ m}$

$\delta_6 = \text{Percentage difference}$

$\delta_{RE} = \text{Error estimate}$

$\varepsilon = \text{Turbulent dissipation rate, m}^2\text{s}^{-3}$

$\varepsilon_{##} = \text{Change between } \hat{\delta}_# \text{ for different grids, example, } \varepsilon_{21} = \hat{\delta}_2 - \hat{\delta}_1$

$\eta = \text{Falkner-Skan parameter, } \gamma \sqrt{\frac{(1+m)U_\infty \{x\}}{2\nu X}}$

$\eta^* = \text{Dimensionless displacement thickness, } \lim_{\eta \to \infty} [\eta - f]$}

$\eta_t = \text{Transition length, m}$

$\gamma = \text{Intermittency based on } C_i$

$\lambda = \text{Thwaites correlation parameter, } \frac{\theta^2 U'_\infty \{x\}}{\nu}$
\[ \mu = \text{Absolute viscosity, } \text{kg/(m s)} \]

\[ \mu_t = \text{Absolute turbulent viscosity, } \rho \nu_t \]

\[ \omega = \text{Specific turbulence dissipation rate, } 1/\text{s} \]

\[ \theta = \text{Momentum thickness, } \int_0^\infty \frac{u}{U_{fs}} \left(1 - \frac{u}{U_{fs}}\right) dy, \text{ m} \]

\[ \nu = \text{Kinematic viscosity, } \mu/\rho, \text{ m}^2/\text{s} \]

\[ \nu_t = \text{Turbulent viscosity, } \text{m}^2/\text{s} \]

\[ \rho = \text{Density, } \text{kg/m}^3 \]

\[ \tau = \text{Wall shear stress, } \mu \frac{\partial u}{\partial y} \bigg|_{y=0} \]

**Superscripts**

\[ (\_)^+ = \text{Normalization by wall units} \]

\[ \_\', \_\'', \_\''' = \text{First, second, and third derivatives, respectively} \]

**Subscripts**

\[ 1, 2, 3 = \text{Corresponds to the fine, medium, or coarse grid, respectively} \]

\[ \text{fs} = \text{Freestream value} \]

\[ i, j, k = \text{Represents variables corresponding to one of the three directions, } x, y, \text{ or } z, \text{ independently.} \]

\[ L = \text{Length of the plate} \]

\[ 0 = \text{Value at the inlet} \]
Acknowledgments

A special thanks to advisor Dr. Tao Xing for significant contributions to the work.
The author thanks Dr. Donald McEligot, Dr. Tamer Zaki, Dr. Kevin Nolan, Dr. John
Crepeau, and Dr. Edmond Walsh for meaningful contributions and Dr. Ralph Budwig and
Dr. Jim Liou for serving on the committee. Gratitude is also extended to the Nuclear
Regulatory Commission for the financial support provided in the form of a fellowship. The
DOE EPSCoR program sponsored this study.
# Table of Contents

Authorization to Submit Thesis ................................................................. ii

Abstract ................................................................................................. iii

Nomenclature ........................................................................................... v

Non-Dimensional Quantities .................................................................... vi

Greek Symbols ........................................................................................ viii

Superscripts ............................................................................................ ix

Subscripts ............................................................................................... ix

Acknowledgments .................................................................................... x

List of Figures .......................................................................................... xv

List of Tables ............................................................................................ xvii

Chapter 1: Introduction .............................................................................. 1

  1.1 Solution Verification Equations .......................................................... 2

Chapter 2: Literature Review .................................................................... 4

  2.1 Analytical .......................................................................................... 4

  2.2 Experimental Studies ......................................................................... 4

  2.3 Direct Numerical Simulation .............................................................. 5

  2.4 RANS Model Simulations .................................................................. 8

  2.5 Large Eddy Simulations ................................................................... 8
References .......................................................................................................................... 59

Appendices .......................................................................................................................... 66

Appendix A. Hydrodynamic stability .................................................................................. 66

Appendix B. Code .................................................................................................................. 68

B.1 Orr-Sommerfeld Solution (Written in Matlab) .............................................................. 68

B.2 Post-processing (Written in SciLab) .............................................................................. 72
List of Figures

Figure 1: Representation of the mesh and geometry .............................................. 13
Figure 2: Residuals for pressure outlet top boundary condition ............................. 16
Figure 3: $\beta$ versus $x/L_x$ .............................................................................. 22
Figure 4: $F(\lambda)$ versus $Re_\theta$ for (A) Thwaites’ correlation (TC) (B) McEligot and
Walsh’s correlation (MC) .................................................................................. 24
Figure 5: $S(\lambda)$ versus $Re_\theta$ for (A) Thwaites’ correlation (TC) (B) McEligot and
Walsh’s correlation (MC) .................................................................................. 25
Figure 6: $C_dRe_\theta$ versus $Re_\theta$ for McEligot and Walsh’s correlation (MC) .......... 26
Figure 7: $F(\lambda)$ versus $\beta$ ............................................................................. 26
Figure 8: $S(\lambda)$ versus $\beta$ ............................................................................. 27
Figure 9: $C_dRe_\theta$ versus $\beta$ ............................................................................. 27
Figure 10: Mean velocity profile at inlet ............................................................... 36
Figure 11: Reynolds normal stress profile at inlet .................................................. 36
Figure 12: Reynolds shear stress profile at inlet ..................................................... 37
Figure 13: $Re_\theta$ versus $Re_x^{1/2}$ ....................................................................... 39
Figure 14: $C_f$ versus $Re_x^{1/2}$ ......................................................................... 41
Figure 15: $C_d$ versus $Re_x^{1/2}$ ......................................................................... 42
Figure 16: $\gamma$ versus $\eta_f$ ................................................................................ 43
Figure 17: $Re_\theta$ versus $Re_x^{1/2}$ ....................................................................... 51
Figure 18: $C_f$ versus $Re_x^{1/2}$ ......................................................................... 52
Figure 19: $C_d$ versus $Re_x^{1/2}$ ......................................................................... 53
Figure 20: $\gamma$ versus $\eta_f$ .......................................................................................................................... 54

Figure 21: DES Shielding Function Contour .......................................................................................... 54

Figure 22: IDDES contour of $\bar{u} \bar{v}'$ at $x/L_x = 0.04$ ........................................................................ 55

Figure 23: IDDES contour of $\bar{u} \bar{v}'$ at $x/L_x = 0.91$ ........................................................................ 56

Figure 24: Temporal eigenfunction ........................................................................................................... 67
## List of Tables

Table 1: Simulation design table ................................................................. 14
Table 2: Variables for both correlations ......................................................... 18
Table 3: Solution verification for laminar boundary layer flow ...................... 21
Table 4: Solution verification for transitional boundary layer flow .............. 38
Chapter 1: Introduction

CFD is one of three approaches to study fluid dynamics; the other two approaches being experimental and analytical. CFD is a powerful research and design tool for many engineering applications, including: manufacturing [6], environmental engineering [7], and naval architecture [8]. Anderson [9] states that CFD is governed by three laws: mass is conserved, Newton’s second law, and energy is conserved. These laws lead to the governing equations of fluid flow: the continuity, momentum, and energy equations. In CFD, the continuous partial differential equations within the Navier-Stokes (NS) equations are discretized into a system of algebraic equations and solved for approximate solutions. Today’s researchers are able to perform large-scale CFD simulations using billions of grid points with the help of modern high performance computing (HPC) techniques.

Entropy is the irreversible loss of energy in fluid flows. Determining and minimizing these losses improves the efficiency of a system [10]. Systems that benefit from the minimization of entropy generation include: cooling systems for electronic devices and nuclear reactors, thermal heat exchangers, and more. Determining the entropy generation in a fluid system is difficult as heat flux and viscous effects lead to the production of entropy. This study focuses on the entropy generated from the viscous aspects of fluid flow.

Steady, unheated, laminar flow entropy generation occurs only from the viscous losses associated with mean velocity gradients. This study hopes to elucidate the mechanisms of entropy generation that occur in laminar flow with both adverse and favorable streamwise pressure gradients. Comparisons are made between different existing
correlations to determine which correlation more closely matches the CFD results for entropy generation in laminar boundary layer flow.

Bypass transition occurs when freestream vortical disturbances induce transition in a boundary layer without the intervention of viscous Tollmien-Schlichting waves [11]. This study models bypass transition using the RANS, LES, and IDDES models. The study hopes to elucidate the boundary layer entropy generation predicted by the different turbulence models. Comparisons are made to the entropy generation predicted by DNS for bypass transitional flows.

1.1 Solution Verification Equations

Solution verification is important to estimate the numerical errors and grid uncertainties of a CFD simulation. Numerical errors are due to the numerical solution of the mathematical equations. Aspects of the simulation that cause numerical errors include: discretization, artificial dissipation, incomplete iterative and grid convergence, and computer round-off. Determining numerical errors generally involves performing a sensitivity study by varying the mesh spacing and/or time step size to a smaller value and evaluating the differences between the solutions. Here $S_1$, $S_2$, $S_3$ represent the fine, medium, and coarse grid solutions of any variable in the simulation, respectively. The relative difference between CFD results and correlation values, represented below as $A$, is calculated as,

$$\delta_{se} = \left| \frac{A - S_i}{A} \right| \times 100\%$$

(1)

The solution verification study requires the use of the following equations from Xing and Stern [12, 13] with the use of L2 norm for profiles from Wilson et al. [14],
\[ \varepsilon_{21} = S_2 - S_1 \]  
\[ \varepsilon_{32} = S_3 - S_2 \]  
\[ \langle R_G \rangle = \|\varepsilon_{21}\|_2 / \|\varepsilon_{32}\|_2 \]  
\[ \langle p_G \rangle = \frac{\ln(\|\varepsilon_{32}\|_2 / \|\varepsilon_{21}\|_2)}{\ln(r_G)} \]  

Monotonic convergence is achieved when \( 0 < \langle R_G \rangle < 1 \).

The solution verification method in place is the factor of safety method. The ratio of the estimated order of accuracy to the theoretical order of accuracy is defined as,

\[ p = \frac{\langle p_G \rangle}{p_{th}} \]  

Usually, the closer \( p \) is to 1, the better quality of CFD results. The grid uncertainty is defined as,

\[ U_G = \begin{cases} [1.6P + 2.45(1-P)]|\delta_{RE}| & 0 < P \leq 1 \\ [1.6P + 14.8(P-1)]|\delta_{RE}| & P > 1 \end{cases} \]  

where \( U_G \) is a percentage of the correlation value or the value from the fine grid solution for the given variable. The error estimate, \( \delta_{RE} \), is defined as,

\[ \delta_{RE} = \frac{\varepsilon_{21}}{r_{PG} - 1} \]  

A lower magnitude of \( U_G \) usually indicates a better quality of CFD results.
Chapter 2: Literature Review

A literature review is performed to expand the extent to which boundary layer entropy generation and bypass transition have been studied using different fluid dynamic approaches.

2.1 Analytical

Entropy generation can be determined analytically for laminar and steady flows. Only the mean velocity and mean heat flux entropy generation mechanisms are taken into account in analytical studies. Ozkol and Arikoglu [15] performed an analytical study on natural convection of laminar flow over a vertical wall of constant temperature. An equation was derived for the minimization of the total entropy generation, and therefore, the optimization of energy convergence systems. Esfahani and Jafarian [16] conducted an analytical study on the total entropy generation within a zero pressure gradient, laminar, flat plate boundary layer over a vertical wall of constant temperature. The study compared three different methods of predicting entropy generation: an integral solution, a similarity solution, and a Blasius series solution. It was shown that all solution methods produce a constant dimensionless total entropy generation but the similarity solution produces less total entropy generation than the other solutions.

2.2 Experimental Studies

Experimental studies are paramount to the understanding boundary layer entropy generation. Sampling transient characteristics of transitional or turbulent flows can make entropy generation approximation difficult in experimental studies. Data can be non-
conditionally sampled, laminar-conditioned, turbulent-conditioned, and intermittency weighted (which is a combination of the two conditionally sampled methods). Experimental results analyzed by Walsh et al. [17] compared the entropy generation approximated from different transient flow sampling methods. The experimental study involved transitional flow over a flat plate with both a zero and a favorable pressure gradient (with $\beta = 0.27$) with $Re_\theta \leq 500$. The intermittency-weighted data predictions were 20% lower than the non-conditionally sampled data for low Reynolds numbers. Nolan et al. [18] analyzed experimental results to develop a semi-empirical technique for predicting entropy generation in transitional boundary layers with zero and favorable pressure gradients. Adeyinka and Naterer [19] post-processed particle image velocimetry (PIV) data of turbulent flows within a channel. The experiment involved flows with Reynolds numbers based on friction velocity, $Re_{\alpha_r}$, from 187-399. The results demonstrated a 3% deviation from the results of White [4]. The study compared with DNS to confirm that the turbulent entropy production was modeled correctly.

### 2.3 Direct Numerical Simulation

DNS is a proven tool in elucidating flow physics. DNS completely resolves all of the laminar and turbulent length scales and thus can be used as a numerical benchmark to evaluate simulations using various turbulence models. McEligot et al. [20] analyzed DNS results from two different studies conducted by Spalart [21, 22] of turbulent boundary layer flows with zero and small favorable pressure gradients with $Re_\theta$ ranging from 300 to 1410. The study found that approximately two-thirds of the entropy generation occurs in the viscous layer of a turbulent boundary layer (defined as $y^+ \approx 30$). The study demonstrated that
entropy dissipation is nearly universal within the viscous layer of turbulent boundary layer flows with zero and small favorable pressure gradients. The study showed that the methodology developed by Rotta [23] for approximating $S''$ is inaccurate for the given flow characteristics. McEligot et al. [24] similarly analyzed results from a DNS [25] of turbulent channel flow with zero and small favorable pressure gradients. The inlet conditions of the channel flow were $Re_{Dh} = 49,000$, where $Dh$ represent the hydraulic diameter of the channel. The computational domain size was $(L_x, L_y, L_z)/y_{c}^+ = (12.8, 2, 6.4)$ where $y_{c}^+$ is the distance to the centerplane with a grid resolution of $(n_x, n_y, n_z) = (1024, 256, 1024)$. The spatial resolution was $(\Delta x^+, \Delta y^+, \Delta z^+) = (8, 0.15-8, 4)$. The study employed a fourth-order central difference scheme for the streamwise and spanwise directions and a second-order central difference scheme in the wall-normal direction. The Crank-Nicholson method was applied for the viscous terms with wall-normal derivatives and the second-order Adams-Bashforth method was employed for other terms. The study compared two methods for approximating entropy generation. The first method evaluated the fluctuating gradients forming the dissipation term in the turbulent enthalpy equation and the second method evaluated an approximate analogy to laminar flow employing assumed boundary layer (and other) approximations [26]. The study demonstrated that both methods predict similar $S''$ values. The second method listed above under-predicted entropy generation in the “linear” layer and over-predicted entropy generation in the rest of the viscous layer. A study by McEligot et al. [27] compared the entropy generation predicted from a DNS of turbulent boundary layer flow to the entropy generation predicted from a DNS of channel flow [25, 28]. The study demonstrated that the pointwise entropy generation at the boundary of the viscous layer is relatively insensitive for both boundary layer and channel flows with large favorable pressure.
gradients. The integral over the area of the viscous layer decreased moderately only for boundary layer flows. Walsh and McEligot [29] improved an existing correlation for the dissipation coefficient, $C_d$, using data from multiple DNS studies of low $Re_\theta$ turbulent boundary layer and channel flows with zero and mild favorable pressure gradients [22, 25, 30, 31]. Walsh et al. [10] analyzed a DNS of bypass transitional boundary layer flows for $Re_\theta$ ranging from 115 to 520 [32, 33]. The study demonstrated that the term for turbulent convection in the turbulent kinetic energy (TKE) balance is dominant within the transition region. This resulted in more turbulent energy being produced than dissipated. The study showed that a popular approximation method over-estimates the dissipation coefficient by as much as 17%. The study demonstrated that the approach developed by Rotta [23] is more accurate for transitional boundary layers. A DNS of bypass transition was performed by Zaki and Durbin [11]. This simulation showed that high-frequency, freestream fluctuations are kept from entering the boundary layer due to “shear sheltering.” The study evaluated the coupling coefficient between continuous spectrum Orr-Sommerfeld and Squire modes. The study demonstrated that a strongly and weakly coupled high-frequency mode is required to completely simulate the transition process.

The bypass transition simulations are compared to a DNS by Nolan and Zaki [1]. The computational domain size was $(L_x, L_y, L_z)/\delta_0 = (900, 40, 30)$ with a grid resolution of $(n_x, n_y, n_z) = (3072, 192, 192)$. The spatial resolution was $(\Delta x^+, \Delta y^+, \Delta z^+) = (11.7, \geq 0.40, 6)$. The inlet mean velocity profile was created based on $Re_{\infty_0} = 800$ with a turbulent intensity of 3%. The study tracked down turbulent spots resulting from high-amplitude streaks upstream. The
study found that the volumetric growth rate of turbulent spots is insensitive to the pressure gradient.

2.4 RANS Model Simulations

More recently, a CFD study by Ghasemi et al. [5] was conducted that evaluated the accuracy of different turbulence models for predicting boundary layer behavior and entropy generation in bypass transition. The models implemented in the study were the k-ε model, the SST k-ω model, the k-ω 4 equation model, the k-kl-ω 3 equation model, and the Reynolds stress model (RSM). The mesh used in the study contained 149,089 grid points on a two-dimensional, zero pressure gradient domain equal to $L_x/\delta_0 = 900$. The inlet boundary condition specified a turbulent intensity of 3% with a turbulent length scale equal to the boundary layer thickness at the inlet. The study showed that the RANS models predict the onset of transition earlier than the corresponding DNS [1]. The RANS models over-predicted the integral entropy generation rate and the skin friction coefficient in the transition region.

2.5 Large Eddy Simulations

The LES model is becoming more frequently implemented in CFD research because it resolves instead of models most of the turbulence length scales like RANS. The use of LES usually leads to more accurate solutions over the RANS models but with a lower computational cost than DNS. Many LES SGS models have been developed in the past decades, allowing researchers to cater the LES model to better capture the physics of the flow. A study by Lardeau et al. [34] focused on unsteady boundary layer processes before, during, and after bypass transitional boundary layer flow over a flat plate. The computational
domain size was \( (L_x, L_y, L_z)/(\delta^e_0) = (400, 35, 40) \) with a grid resolution of \( (n_x, n_y, n_z) = (512, 80, 96) \). The inflow conditions were based on \( Re_\theta = 425 \) with three different inlet turbulent intensities: 2.5\%, 4.3\%, and 8\%. The study applied the localized Lagrangian-averaged dynamic eddy-viscosity SGS model. A second-order, central-difference scheme was applied for the spatial derivatives and a second-order time approximation was applied for the time integration. The study demonstrated how the pre-transitional, elongated, streaky structures led to the amplification of fluctuations by conventional shear-stress/shear-strain interaction instead of by pressure diffusion. The study noted that uncertainties arise from the lack of realism in the freestream conditions. A study by Lardeau et al. \cite{35} compared the ability of different SGS models to predict separation bubbles in transitional flow over both, a flat plate and a compressor blade. The chord length of the compressor blade was specified as the reference length, \( L_c \). The computational domain size for the flat plate simulation was \( (L_x, L_y, L_z)/L_c = (2, 0.25, 0.12) \) with a plate length of 1.5 \( L_c \). The compressor passage computational domain had a blade-to-blade distance of 0.592 \( L_c \). The grid resolution was \( (n_x, n_y, n_z) = (512, 128, 64) \) and (512, 192, 64) for the flat plate and compressor blade, respectively. The inflow conditions were \( Re_{L_c} = 60,000 \) and \( Re_{L_c} = 138,500 \) for the flat plate and compressor blade, respectively. The inflow turbulent intensities were 0\%, 1\%, 1.5\%, and 2\% for the flat plate and 0\% and 3.25\% for the compressor blade. The study applied the Smagorinsky-Lilly dynamic model, the mixed-time-scale model, and the wall-adapted local eddy viscosity (WALE) model to represent the SGS processes. A second-order, central-difference scheme was applied for the spatial derivatives and a third-order time approximation was applied for the time integration. The study showed that results from all three SGS models are consistent with previous theoretical and numerical analyses for the flat plate geometry. For the
compressor blade geometry, only the mixed-time-scale model returned a fair representation of both pressure and skin friction following separation. The study demonstrated that the sensitivity of the results to the SGS model decreased as the inlet turbulent intensity increased. Monokrousos et al. [36] applied a linear model-based feedback control to delay the onset of transition in bypass transitional flows over a flat plate. The domain sizes in the study were \((L_x, L_y, L_z)/(\delta^*)_0 = (1000, 60, 50), (2000, 60, 90), (2000, 60, 180),\) and \((4000, 60, 180)\) with grid resolutions of \((n_x, n_y, n_z) = (256, 121, 36), (512, 121, 64), (512, 121, 128),\) and \((1024, 121, 128),\) respectively. The inflow conditions were based on \(Re_{\delta^*} = 300\) with an inlet freestream turbulent intensity of 4.7%. The SGS model was ADM-RT. A third-order Runge-Kutta scheme was applied for the time integration scheme and a second-order, Crank-Nicolson scheme was applied for the spatial derivatives. The study demonstrated that the control mechanism was successfully able to reduce the energy of the streaks, preventing the streaks from diffusing into the shear layer near the wall, subsequently delaying the onset of transition.

Limited research has been conducted in the area of boundary layer entropy generation estimation using LES. A study by Sheikhi et al. [37] employed a modification to a filtered density function (FDF) model to determine the entropy generation in turbulent mixing layer flows. The FDF model was modified to close the filtered entropy transport equation with a system of stochastic differential equations. The domain was a cubic box of dimensions \((L_x, L_y, L_z)/(2^{Nv} \lambda_u) = (L_r, L_r, L_r)\) where \(L_r\) is half the initial vorticity thickness, \(Nv\) is the desired number of successive vortex pairings, and \(\lambda_u\) is the wavelength of the most unstable mode corresponding to the mean streamwise velocity imposed at the initial time. The flow was
characterized by two parallel streams of equal velocities entering the domain in opposite directions. A fourth-order scheme was applied for the spatial and time discretization. The study demonstrated that the modifications to the FDF model agree with the DNS data from Huai et al. [38].

In order to accurately capture the flow physics, the LES time and length scales must be sufficiently small. The smallest length scale is the Kolmogorov scale where the viscous dissipation takes place. Since the grid size relates to the filter size, a certain grid size is necessary to capture the energy within different wavenumbers. Ferziger [39] notes that a second-order numerical discretization scheme cannot compute the derivatives of modes with wavenumbers higher than \( k_{\text{max}} = \pi/2\Delta x \), where \( k_{\text{max}} \) is the maximum wavenumber of the Fourier modes. This means that most of the energy should be contained at wavenumbers below \( k_{\text{max}} \). A test case was considered by Kornhaas et al. [40] to determine the appropriate time step size and convergence criterion per time step for a periodic flow over a two-dimensional hill. The domain size was \((L_x, L_y, L_z)/h = (9, 3.03, 4.5)\), where \( h \) is the height of the hill. The simulation had \( 1.47 \times 10^6 \) grid points. The Reynolds number, \( Re_h \), at the inlet was 11,600. The Smagorinsky dynamic model was employed to compute the sub-grid stresses. A second-order central difference scheme was applied spatially and the implicit Crank-Nicolson scheme was applied for time discretization. The study found that LES simulations require a time step corresponding to \( \text{CFL} = 2 \) with a convergence criterion per time step of \( 10^{-2} \).
Chapter 3: Simulation Design

3.1 Laminar Pressure Gradient Geometry and Flow Conditions

The streamwise pressure gradient parameter, $\beta$, represents the change in pressure along the streamwise direction, $dp/dx$. A constant streamwise pressure gradient is created using the shape of the top slip wall. When $\beta < 0$, the pressure gradient is negative or adverse and when $\beta > 0$, the pressure gradient is positive or favorable. A zero pressure gradient occurs when $\beta = 0$. A different constant $\beta$ value is set for each simulation. A non-zero value is set for the streamwise location at the inlet, $x_0$, the inlet freestream velocity, $U_{fs}\{x_0\}$, the height of the wall at the inlet, $h\{x_0\}$, and the volumetric flow rate, $Q$. These values are chosen arbitrarily and remain constant for all laminar simulations. The value for the Falkner-Skan power law parameter, $m$, is determined for each $\beta$ value using,

$$m = \frac{\beta}{2 - \beta}$$  \hspace{1cm} (9)

From this, the Falkner-Skan constant, $K$, is determined using $x_0$ and $U_{fs}\{x_0\}$ in,

$$K = \frac{U_{fs}\{x_0\}}{x_0^m}$$  \hspace{1cm} (10)

This value for $K$ is constant for each value of $\beta$. The freestream velocity profile for the length of the plate is calculated using the Falkner-Skan relation,

$$U_{fs}\{x\} = Kx^m$$  \hspace{1cm} (11)

The cross sectional area determines the height of the top wall in the $y$ direction, $h\{x\}$. The Falkner-Skan [26, 41-43] equation,
\[ 0 = f'''' + ff'' + \beta \left(1 - f'^2\right) \]  

(12)

must be solve to determine the dimensionless displacement thickness. The displacement boundary layer thickness is calculated as,

\[ \delta^* = \frac{\eta^*}{\sqrt{\frac{(1+m)U_{\infty}(x)}{2v_x}}} \]  

(13)

and additionally expands the wall. The displacement thickness is part of the variables that can be calculated for Falkner-Skan flows. The final equation for the shape of the top wall is,

\[ h(x) = \frac{Q}{U_{\infty}(x)} + \delta^* \]  

(14)

Figure 1 is a representation of an adverse pressure gradient geometry and mesh. The coordinate axis and boundaries are labeled.

![Figure 1: Representation of the mesh and geometry](image)
3.2 Meshing

The mesh is created in the meshing software Pointwise v17.0R1. The grid points in the streamwise and span-wise direction (where applicable) are uniform and the grid points in the plate-normal direction are skewed toward the plate, as shown in Figure 1. Skewing the grid toward the wall ensures that enough grid points exist within the boundary layer to fully capture the high velocity gradients near the plate surface. A medium and coarse mesh is created for the solution verification study using a constant grid refinement ratio applied to the fine mesh. The three systematically refined grids have a grid refinement ratio of 2 for the laminar simulations and $2^{\frac{3}{2}}$ for the RANS simulations.

3.3 Simulation Design Table

A general overview of the different simulations performed in this study is contained in Table 1 below.

<table>
<thead>
<tr>
<th>Flow Type</th>
<th>Viscous Model</th>
<th>$L_x \times L_y \times L_z$ (m)</th>
<th>$n_x \times n_y \times n_z$</th>
<th>Number of grid points</th>
<th>$y^+$</th>
<th>Inlet/Outlet Boundary Condition</th>
<th>Time Dependence</th>
</tr>
</thead>
<tbody>
<tr>
<td>Laminar</td>
<td>Laminar</td>
<td>28x8x0</td>
<td>1,027x827x0</td>
<td>8.5x10^5</td>
<td>0.03</td>
<td>Pressure Inlet/Pressure Outlet</td>
<td>Steady</td>
</tr>
<tr>
<td>2D Bypass Transition</td>
<td>k-\epsilon, SST k-\omega, RSM, 4 Eq. SST k-\omega</td>
<td>24.4x1x0</td>
<td>3,163x317x0</td>
<td>1x10^6</td>
<td>0.05</td>
<td>Velocity Inlet/Outflow</td>
<td>Steady</td>
</tr>
<tr>
<td>3D Bypass Transition</td>
<td>IDDES</td>
<td>24.4x1x0.78</td>
<td>570x90x30</td>
<td>1.54x10^6</td>
<td>0.82</td>
<td>Velocity Inlet/Outflow</td>
<td>Unsteady</td>
</tr>
<tr>
<td></td>
<td>LES</td>
<td>24.4x1x0.78</td>
<td>570x90x30</td>
<td>1.54x10^6</td>
<td>0.82</td>
<td>Velocity Inlet/Outflow</td>
<td>Unsteady</td>
</tr>
</tbody>
</table>

Table 1: Simulation design table
3.4 Laminar Parametric Studies

Since there are many ways to setup the simulations, multiple parametric studies are conducted to determine the most appropriate conditions in terms of stability and accuracy. The parametric studies are conducted for the laminar boundary layer simulations for different inlet, outlet, top, and bottom boundary conditions. The inlet boundary conditions in the parametric study include velocity inlet and pressure inlet in ANSYS FLUENT. Due to the fictitious origin of the Falkner-Skan equations, the pressure inlet shows a more accurate pressure gradient across the flow field. Little discernible difference is seen between the outflow and pressure outlet boundary conditions. The pressure outlet condition is chosen because the outflow condition cannot be implemented with the pressure inlet condition. The top boundary condition is evaluated as a pressure inlet, pressure outlet, velocity inlet, and slip wall condition. The slip wall is the only top boundary condition to create the proper pressure gradient and remain stable throughout the solution. The pressure outlet and pressure inlet top boundary conditions are unstable. The instability of the pressure outlet top boundary condition is demonstrated in the residual plot shown in Figure 2.
Figure 2: Residuals for pressure outlet top boundary condition

A leading and trailing slip wall around the no-slip plate is tested as the bottom boundary condition causing a high and low pressure region to occur at the plate’s leading and trailing edge, respectively. Ultimately, no leading or trailing slip walls are part of any simulations due to these pressure bubbles.

3.5 High Performance Computing

Simulations are conducted on different computers: a Dell OptiPlex 990 computer with an Intel® Core™ i7-2600 CPU with 4 cores and 8 GB’s of RAM and a Dell Precision T7500 with 12 cores and 48 GB’s of RAM. Results are post-processed using Tecplot 360 2013.
Chapter 4: Laminar Boundary Layer Flows

4.1 Objective and Approach

The objective of this study is to elucidate the entropy generation within a laminar boundary layer with and without a streamwise pressure gradient. The flow considered is steady, unheated, incompressible, two-dimensional laminar boundary layer flow over a flat plate. The simulations use the commercial CFD software ANSYS FLUENT to determine the boundary layer characteristics. The pressure gradient is applied using a curved top slip wall. The results are compared against the Blasius [44] and Falkner-Skan [41, 43] solutions. The CFD results are used to evaluate the accuracy of the correlation for boundary layer parameters developed by McEligot and Walsh [2]. This correlation is compared to the correlation developed by Thwaites [3, 4]. Quantitative solution verification is conducted using three systematically refined structured grids, with the finest grid containing 1 million grid points.

4.2 Terminologies

The correlations by McEligot and Walsh are modifications from a previous correlation develop by Walz [45]. Table 2 lists the parameter values in the equations below for both correlations.
Table 2: Variables for both correlations

<table>
<thead>
<tr>
<th>Coefficients</th>
<th>Thwaites</th>
<th>McEligot and Walsh</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a$</td>
<td>0.45</td>
<td>0.44105</td>
</tr>
<tr>
<td>$b$</td>
<td>6</td>
<td>5.309</td>
</tr>
<tr>
<td>$\lambda_0$</td>
<td>-0.09</td>
<td>-0.068</td>
</tr>
<tr>
<td>$C_s$</td>
<td>2</td>
<td>2.1348</td>
</tr>
<tr>
<td>$E_s$</td>
<td>0.62</td>
<td>0.58706</td>
</tr>
</tbody>
</table>

The correlations estimate the momentum thickness as,

$$\theta = \sqrt{\frac{avx}{((b-1)m+1)Kx^m}}$$ (15)

the boundary layer correlation function, $F(\lambda)$, as,

$$F(\lambda) = a - b\lambda$$ (16)

and the wall shear stress correlation function as,

$$S(\lambda) = \frac{C_s(\lambda - \lambda_0)^{E_s}}{2}$$ (17)

The relationship between entropy generation and viscous dissipation is discussed by Bejan [46, 47] as well as others. Bejan uses a force balance in the longitudinal direction in a steady pipe flow with friction example to discuss how the entropy generation rate and the streamwise velocity gradient are related. The force balance shows how the mechanical power pushing the flow through the pipe is proportional to the change in pressure across the pipe, which is proportional to the shear stress between the wall and the fluid. The first and second laws of thermodynamics for a steady state system dictate that the entropy generation rate is equal to the rate of work over the initial temperature. The third derivative of the rate of work is the volumetric rate of mechanical power dissipated in an infinitesimally small
Taking the third derivative of the entropy generation rate provides the pointwise entropy generation rate. Substituting the mechanical power dissipated into this equation shows,

\[ TS'''(y) \approx \mu \left( \frac{\partial u}{\partial y} \right)^2 \] (18)

The pointwise entropy generation rate equation applies for steady, two-dimensional, laminar boundary layer flows without significant fluctuations. For laminar flow, only the viscous dissipation for the mean velocity profile contributes to the entropy generation [10]. The gradient of the plate-normal velocity component is negligibly small, and therefore disregarded. The integral over the boundary layer of the pointwise entropy generation rate provides the entropy generation rate per unit area,

\[ TS'' = \int_0^\delta S''' dy \] (19)

The dissipation coefficient, \( C_d \), is a dimensionless variable that represents the entropy generation rate per unit area. The correlation by McEligot and Walsh estimate the dissipation coefficient multiplied by \( Re_\theta \) as,

\[ C_d Re_\theta = 0.1740 + 0.3315\lambda + 0.7881\lambda^2 \] (20)

### 4.3 Simulation Setup

A pressure outlet boundary condition with zero gage pressure is applied at the domain outlet. The top wall is set as a zero shear stress or slip wall boundary condition with a curved shape to create the desired pressure gradient. Since the specification for \( U_{fs}(x_0) > 0 \), a
singularity of the solution of the Falkner-Skan equations exists at the leading edge or (0,0) [4]. This singularity requires $x_0 > 0$. To ensure the proper streamwise pressure gradient occurs, the laminar boundary layer flow simulations use a pressure inlet boundary condition where total pressure is specified at the inlet. The total inlet pressure is determined using the static gage pressure throughout the flow field. The static gage pressure is determined for each $x$ location using Bernoulli’s equation,

$$
-dp/dx = \rho U_{fs}(x) dU_{fs}(x)/dx
$$

When Eq. (11) is inserted into Eq. (21), and is integrated by $x$, the equation becomes,

$$
 p_t = p_t + \frac{1}{2} \rho \left( Kx^m \right)^2
$$

where the total gage pressure, $p_t$, is set to the pressure at the lowest value of $U_{fs}(x)$. The lowest value of $U_{fs}(x)$ occurs at the inlet for favorable pressure gradients and the outlet for adverse pressure gradients.

The third-order momentum upstream-centered scheme for conservation laws (MUSCL) scheme is applied for the momentum solver and a second-order scheme is applied for the pressure solver. The semi-implicit method for pressure linked equations (SIMPLE) is applied as the pressure-velocity coupling scheme. A convergence tolerance of $1 \times 10^{-10}$ is set for the simulations. A low convergence tolerance ensures that the iterative errors are much smaller than the grid errors such that the former can be neglected.
4.4 Solution Verification

The solution verification results for all values of \( \beta \) are presented in Table 3, with \( S_1 \), \( S_2 \), and \( S_3 \) being the fine, medium, and coarse grid solutions, respectively, for the variable in the first row of the table. Table 3 shows that all simulations achieve monotonic convergence since all the values for \( R_G \) are between 0 and 1. The asymptotic range is reached as the discretization parameter becomes asymptotically “close to zero.” If the distance metric to the asymptotic range, \( P \), is 1, then the solutions are in the asymptotic range. Table 3 shows that \( C_d \) is the farthest from the asymptotic range with a maximum \( P \) value of only 0.325. The value of \( P \) for the most favorable pressure gradient (\( \beta = 0.5 \)) is higher than the same value for the most adverse pressure gradient (\( \beta = -0.14 \)). When \( \beta = 0.25 \), \( P \) for the variable \( F[\lambda] \) is 68% lower than the highest value for \( P \) and the grid uncertainty is 90% higher than the lowest grid uncertainty. The grid uncertainty is below 1% for six of the seven values of \( \beta \). The dissipation coefficient grid uncertainty remains below 0.005% for all beta values.

Overall, the grid uncertainties are lower than 1.5% for all variables at all values of \( \beta \). The solution verification study demonstrates that the CFD results are independent of the grid resolution.

<table>
<thead>
<tr>
<th>( \beta )</th>
<th>( R_G )</th>
<th>( P )</th>
<th>( U_G(%S_1) )</th>
<th>( R_G )</th>
<th>( P )</th>
<th>( U_G(%S_1) )</th>
<th>( R_G )</th>
<th>( P )</th>
<th>( U_G(%S_1) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>0.371</td>
<td>0.477</td>
<td>0.618</td>
<td>0.238</td>
<td>0.690</td>
<td>0.034</td>
<td>0.509</td>
<td>0.325</td>
<td>0.001</td>
</tr>
<tr>
<td>0.25</td>
<td>0.679</td>
<td>0.186</td>
<td>1.499</td>
<td>0.284</td>
<td>0.606</td>
<td>0.085</td>
<td>0.604</td>
<td>0.242</td>
<td>0.002</td>
</tr>
<tr>
<td>0.05</td>
<td>0.347</td>
<td>0.509</td>
<td>0.153</td>
<td>0.279</td>
<td>0.614</td>
<td>0.104</td>
<td>0.690</td>
<td>0.178</td>
<td>0.004</td>
</tr>
<tr>
<td>0</td>
<td>0.310</td>
<td>0.564</td>
<td>0.144</td>
<td>0.297</td>
<td>0.583</td>
<td>0.131</td>
<td>0.707</td>
<td>0.167</td>
<td>0.004</td>
</tr>
<tr>
<td>-0.05</td>
<td>0.320</td>
<td>0.548</td>
<td>0.189</td>
<td>0.292</td>
<td>0.593</td>
<td>0.124</td>
<td>0.727</td>
<td>0.153</td>
<td>0.004</td>
</tr>
<tr>
<td>-0.1</td>
<td>0.289</td>
<td>0.597</td>
<td>0.197</td>
<td>0.268</td>
<td>0.633</td>
<td>0.129</td>
<td>0.735</td>
<td>0.148</td>
<td>0.004</td>
</tr>
<tr>
<td>-0.14</td>
<td>0.368</td>
<td>0.481</td>
<td>0.327</td>
<td>0.546</td>
<td>0.378</td>
<td>0.670</td>
<td>0.708</td>
<td>0.166</td>
<td>0.004</td>
</tr>
</tbody>
</table>
4.5 Results

Figure 3 compares the $\beta$ values from the Falkner-Skan equations. The CFD $\beta$ values and the Falkner-Skan $\beta$ values agree very well except near the inlet. The discrepancy at and near the inlet is due to the unphysical nature of the inlet boundary condition. The pressure inlet boundary condition is chosen due to the singularity of the Falkner-Skan equations at $x_0 = 0$. Data is only collected from the region of $x/L_x > 60\%$ to avoid issues with the inlet boundary condition.

![Figure 3: $\beta$ versus $x/L_x$](image)

Figure 4 to Figure 6 show the values for the $F(\lambda)$, $S(\lambda)$, and $C_dRe_0$ in the streamwise direction. All three figures show that the more adverse the pressure gradient becomes, the less the computational results remain constant in the streamwise direction. The figures show
that as the magnitude of $\beta$ increases, the difference between the two correlations increases. This trend is more explicit for the adverse pressure gradients than for the favorable pressure gradients. The discrepancies at larger magnitudes of $\beta$ could be due to the limitations of the Falkner-Skan solutions. The correlations only apply to $-0.1987 < \beta < 2$. This is because separation occurs as the pressure gradient becomes more adverse and the boundary layer behavior in extremely favorable pressure gradients is not consistent with the Falkner-Skan predictions.

Figure 7 to Figure 9 show the values for $F\{\lambda\}$, $S\{\lambda\}$, and $C_dRe_\theta$ for different values of $\beta$. These values are all evaluated at $x/L_x = 86\%$. As the value of $\beta$ increases from -0.14 to 0.5, the cross-sectional area at the streamwise location decreases and the freestream velocity increases, resulting in a thinner boundary layer with a larger velocity gradient at the plate surface. This mechanism is the reason the boundary layer growth is larger for adverse pressure gradients as shown in Figure 7. The values of $S\{\lambda\}$, and $C_dRe_\theta$ increase and the value of $F\{\lambda\}$ decreases as $\beta$ increases.

The bar graphs on the bottom of Figure 7 to Figure 9 show the $\delta_{\%}$ between the correlations and the CFD results. The figures show that $\delta_{\%}$ increases as the magnitude of $\beta$ increases. The percentage difference between the Thwaites’ correlation and the CFD results exceeds 5% at $\beta$ values -0.14, -0.1, 0.25. and 0.5 for $F\{\lambda\}$ and $S\{\lambda\}$. The percent difference between McEligot and Walsh’s correlation and the CFD results remains under 2.5% for all $\beta$ values except at the largest adverse pressure gradient.
Figure 4: $F(\lambda)$ versus $Re_\theta$ for (A) Thwaites’ correlation (TC) (B) McEligot and Walsh’s correlation (MC)
Figure 5: $S(\lambda)$ versus $Re_\theta$ for (A) Thwaites’ correlation (TC) (B) McEligot and Walsh’s correlation (MC)
Figure 6: $C_d R e_\theta$ versus $Re_\theta \beta$ for McEligot and Walsh’s correlation (MC)

Figure 7: $F(\lambda)$ versus $\beta$
Figure 8: \( S(\lambda) \) versus \( \beta \)

Figure 9: \( C_d Re_0 \) versus \( \beta \)
4.6 Conclusions

The laminar CFD results demonstrate that entropy generation in the boundary layer increases as the pressure gradient becomes more favorable. The boundary layer growth across the plate increases as the pressure gradient becomes more adverse. The correlations predict constant values for the variables $F(\lambda), S(\lambda)$, and $C_d Re_\theta$ in the streamwise direction, however the CFD results show an up to 1.5% variable variation along the flat plate under strong adverse pressure gradients. These variations are likely due to the assumption that the plate-normal velocity component is negligible. The correlations neglect this aspect of the flow while the CFD results would not. Figure 4 to Figure 6 show the CFD results under predict the values of $F(\lambda)$ and over predict the values of $S(\lambda)$ and $C_d Re_\theta$ in the streamwise direction. These variations could be the result of comparing the Falkner-Skan solutions with the NS solutions for the flow field. This study demonstrates that the correlation developed by McEligot and Walsh more accurately predicts the $F(\lambda), S(\lambda)$, and $C_d Re_\theta$ of laminar flow over a flat plate with streamwise pressure gradients. The predictions from the McEligot and Walsh correlation for $C_d Re_\theta$ agree well with the CFD data.
Chapter 5: Bypass Transition Boundary Layer Flows Using Reynolds Averaged Navier-Stokes

5.1 Objective and Approach

The objective of this study is to evaluate the ability of various RANS turbulence models to predict entropy generation and flow behavior within a bypass transitional boundary layer. The flow considered is steady, unheated, incompressible, two-dimensional bypass transitional boundary layer flow over a flat plate. Various turbulence models within the commercial CFD software ANSYS FLUENT are employed to determine the boundary layer characteristics. The turbulence models employed in the study are the k-\( \varepsilon \) model, k-\( \omega \) SST model, transitional 4 equation SST k-\( \omega \) model, and the RSM. The flow characteristics are compared to the DNS results from Nolan and Zaki [1] and a recent CFD study by Ghasemi et al. [5].

5.2 Governing Equations

The non-linear Reynolds stress term is closed in the RANS models with the Boussinesq eddy viscosity hypotheses [48]. The hypothesis calculates the Reynolds stresses as,

\[
\overline{u_i u_j} = \frac{2}{3} k \delta_{ij} - \nu_i \left( \frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right)
\]

(23)

where the variable \( u_i \) is the velocity along the \( x, y, \) or \( z \) axis and \( u_j \) is the velocity along an axis different from the direction of \( u_i \). This similarly applies to \( x_i \) as the location along a given axis \( x, y, \) or \( z \). The variable \( \delta_{ij} \) in the equation is the Kronecker delta and not the
boundary layer thickness. The turbulent viscosity is calculated differently depending on the model in use.

The transport equations for the different models can be summarized as,

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

(24)

\[
\frac{\partial}{\partial t} \left( \rho [DV] \right) + \frac{\partial}{\partial x_i} (\rho [DV] u_i) = \frac{\partial}{\partial x_j} \left( \left( \mu + \frac{\mu_t}{\sigma_{D'}} \right) \frac{\partial [DV]}{\partial x_j} \right) + G_D - Y_D + D_D
\]  

(25)

where \([DV]\) is the corresponding turbulence dissipation variable for the model.

5.2.1 k-\(\varepsilon\) Model

The formulations for the k-\(\varepsilon\) model are described in the ANSYS FLUENT Theory Guide [49]. The transport equations for the k-\(\varepsilon\) model require the following,

\[
[DV] = \varepsilon; \quad G_k = -\rho u' \overline{u_j} \frac{\partial u_j}{\partial x_i}; \quad Y_k = \rho \varepsilon; \quad G_D = 1.44 \frac{\varepsilon}{k} G_k;
\]

\[
Y_D = 1.92 \rho \frac{\varepsilon^2}{k}; \quad D_D = 0; \quad \sigma_k = 1.0; \quad \sigma_{D'} = 1.3
\]  

(26)

where the turbulent viscosity is calculated as,

\[
\mu_t = 0.9 \rho \frac{k^2}{\varepsilon}
\]  

(27)

5.2.2 Shear Stress Transport k-\(\omega\) Model

The formulations for the SST k-\(\omega\) model are described in the ANSYS FLUENT Theory Guide [49]. The transport equations for the SST k-\(\omega\) model require the following,
\[
[DV] = \omega, G_k = \min(-\rho u_i u'_j \frac{\partial u_j}{\partial x_i}, 10Y_k),
\]

\[
Y_k = 0.09 \rho k \omega, G_D = \frac{\alpha}{\nu} G_k, Y_D = \frac{\rho \beta \omega^2}{1.168}
\]

\[
\sigma_k = \frac{1}{F_i / 1.176 + (1 - F_i) / 1.0}
\]

\[
\sigma_D = \frac{1}{F_i / 2.0 + (1 - F_i) / 1.168}
\]

and the turbulent viscosity is calculated as,

\[
\mu_t = \frac{\partial k}{\partial \omega} \frac{1}{\max(1, \frac{S_f F_z}{0.31 \omega})}
\]

In these equations, not all the variables are constants as is the case for the k-\varepsilon model. The following lists the formulas for the variables within the transport and turbulent viscosity equations,

\[
\alpha = 0.553 F_1 + 0.440 (1 - F_1)
\]

\[
\beta = 0.075 F_1 + 0.0828 (1 - F_1)
\]

where \(F_1\) and \(F_2\) are blending functions defined as,

\[
F_i = \tanh(\Phi_i^4)
\]

\[
\Phi_i = \min \left[ \max \left( \sqrt{\frac{k}{0.09 \omega y}}, \frac{500 \nu}{y^2 \omega}, \frac{4\rho k}{1.168 D^*_a y^2} \right), \frac{4\rho k}{1.168 D^*_a y^2} \right]
\]

\[
D^*_a = \max \left[ 2\rho \frac{1}{1.168 \omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_j}, 10^{-10} \right]
\]
\( F_2 = \tanh(\Phi_2^2) \)
\[
\Phi_2 = \max \left[ 2 \sqrt{\frac{\sqrt{k}}{0.09\omega y}} \cdot \frac{500\nu}{y^2\omega} \right]
\]

(33)

5.2.3 Transition Shear Stress Transport Model

The formulations for the transition SST model are described in the ANSYS FLUENT Theory Guide [49]. The transition SST model couples two additional transport equations with the SST k-\( \omega \) transport equations. The first additional transport equation is for the intermittency and is defined as,

\[
\frac{\partial}{\partial t} (\rho \gamma) + \frac{\partial}{\partial x_j} (\rho u_j \gamma) = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_t} \frac{\partial \gamma}{\partial x_j} \right) + P_{\gamma_1} - E_{\gamma_1} + P_{\gamma_2} - E_{\gamma_2}
\]

(34)

The transition sources are defined as,

\[
P_{\gamma_1} = 2F_{\text{length}} \rho S_T (\gamma F_{\text{onset}})^{0.5}
\]
\[
E_{\gamma_1} = P_{\gamma_1} \gamma
\]

(35)

where \( F_{\text{length}} \) is an empirical correlation that controls the length of the transition region. The destruction/relaminarization sources are defined as,

\[
P_{\gamma_2} = 0.06 \rho \Omega \gamma F_{\text{turb}}
\]
\[
E_{\gamma_2} = 50 \gamma P_{\gamma_2}
\]

(36)

where \( \Omega \) is the vorticity magnitude. The onset of transition is controlled by,

\[
Re_\gamma = \frac{\rho y^2 S_T}{\mu}, R_t = \frac{\rho k}{\mu \omega}
\]

(37)
\[ F_{\text{onset}1} = \frac{Re_c}{2.193Re_{\theta c}} \]
\[ F_{\text{onset}2} = \min\left(\max\left(\frac{Re_c}{F_{\text{onset}1}^4}, 2.0\right), 2.0\right) \]
\[ F_{\text{onset}3} = \max\left(1 - \frac{Re_c}{2.5}, 0\right) \]
\[ F_{\text{onset}} = \max\left(F_{\text{onset}2} - F_{\text{onset}3}, 0\right) \]
\[ F_{\text{turb}} = \epsilon \left(\frac{Re_c}{4}\right)^4 \]

where \( S_f \) is the strain rate magnitude and \( Re_{\theta c} \) is the critical Reynolds number where the intermittency first starts to increase in the boundary layer. The transport equation for the transition momentum thickness Reynolds number, \( Re_{\theta t} \) is,

\[
\frac{\partial}{\partial t}\left(\rho Re_{\theta t}\right) + \frac{\partial}{\partial x_j}\left(\rho u_j Re_{\theta t}\right) = \frac{\partial}{\partial x_j}\left(2\left(\mu + \mu_t\right) \frac{\partial Re_{\theta t}}{\partial x_j}\right) + P_{\theta t}
\]

with

\[
P_{\theta t} = 0.03 \frac{\mu_0 U^2}{500 \mu} \left(Re_{\theta p} - Re_{\theta t}\right)\left(1.0 - F_{\theta t}\right)
\]

where \( Re_{\theta p} \) is a proprietary empirical correlation for the transition onset and \( F_{\theta t} \) is a function based on the boundary layer correlations calculated through FLUENT.

### 5.2.4 Reynolds Stress Model

The formulations for the RSM are described in the ANSYS FLUENT Theory Guide [49]. The transport equation for the RSM is,
\[
\frac{\partial}{\partial t}(\rho u_i u_i') + \frac{\partial}{\partial x_k}(\rho u_i u'_k) = -\frac{\partial}{\partial x_k}\left(\rho \delta_{ij} u_i' u_i' + p \left(\delta_{ij} u_i' + \delta_{ij} u_i'\right)\right)
\]
\[
+ \frac{\partial}{\partial x_k}\left(\mu \frac{\partial u_i'}{\partial x_k}\right) - \rho \left(u_i' \frac{\partial u_i}{\partial x_k} + u_i' \frac{\partial u_i}{\partial x_i}\right) + \frac{\partial u_i'}{\partial x_i} + \frac{\partial u_i'}{\partial x_i}
\]
\[
-2\mu \frac{\partial^2 u_i'}{\partial x_k^2} = 2\rho \Omega_k \left(u_j' u_m' \epsilon_{ikm} + u_j' u_m' \epsilon_{jkm}\right)
\]

where \(\epsilon_{ikm}\) and \(\epsilon_{jkm}\) are permutation symbols.

### 5.3 Post-Processing

Fluctuations in bypass transitional flows necessitate additional terms to the entropy generation equations used for laminar flow. These equations are outlined further in Walsh et al. [10]. The dimensionless entropy generation rate per unit area is calculated as,

\[
(S'\{\delta\})^+ \approx \int_0^\delta \left(\frac{\partial U^+}{\partial y^+}\right)^2 dy^+
\]
\[
-\int_0^\delta \left[\left(u'^2\right)^+ - \left(v'^2\right)^+\right] \frac{\partial U^+}{\partial x^+} dy^+
\]
\[
- \frac{d}{dx^+} \left[\delta U^+ \left(q^2\right)^+ dy^+\right]
\]

The dimensionless form of Eq. (42) is the dissipation coefficient,

\[
C_d = \left(S'\{\delta\}\right)^+ \left(\frac{C_f}{2}\right)^{3/2}
\]

Intermittency is a measure of determining the laminar, transition, and turbulent regions of the flow. Intermittency is calculated as,

\[
\gamma = \frac{(C_f - C_{f,\text{lam}})}{(C_{f,turb} - C_{f,\text{lam}})}
\]
Where the skin friction coefficient variables for the laminar and turbulent regions are calculated respectively as,

\[ C_{f,\text{laminar}} = 0.664 / Re_x^{0.5} \] (45)

\[ C_{f,\text{turbulent}} = 0.455 / \ln^2 (0.06Re_x) \] (46)

The intermittency is compared to transition length,

\[ \eta_i = \frac{x - x_s}{x_e - x_s} \] (47)

where the \( x \) value for the beginning of transition, \( x_s \), is when \( \gamma = 0.005 \) and the \( x \) value for the end of transition, \( x_e \), is when \( \gamma = 0.995 \).

### 5.4 Simulation Setup

The boundary conditions and geometry in the bypass transitional simulations are designed to match the conditions of the DNS by Nolan and Zaki [1]. A velocity inlet is applied to the inlet boundary condition. The mean inlet velocity profile is a Blasius velocity profile at \( Re_\delta = U_{\infty} \delta_0 / \nu = 800 \). The inlet turbulence is based on the mean Reynolds stresses from the DNS. The dimensionless inlet profiles are shown in Figure 10 to Figure 12. The outflow boundary condition is applied to the outlet boundary. A no-slip wall boundary condition is applied to the plate and a slip wall boundary condition is applied to the top wall. The length of the plate, \( L_x \), is the same as the DNS, i.e. \( L_x / \delta_0 = 900 \). The top wall shape matches the geometry provided by Nolan and Zaki from the DNS.
Figure 10: Mean velocity profile at inlet

Figure 11: Reynolds normal stress profile at inlet
The Reynolds stresses are used to determine the appropriate inlet turbulent conditions for each model. The equations for the inlet conditions come from the FLUENT User’s Manual. The $k$-$\varepsilon$ model estimates $\varepsilon$ at the inlet as,

$$
\varepsilon = \frac{0.09^{3/4} k^{3/2}}{0.4\delta_0} \tag{48}
$$

and for the $k$-$\omega$ model, $\omega$ is estimated at the inlet as,

$$
\omega = \frac{0.09^{-1/4} \sqrt{k}}{0.4\delta_0} \tag{49}
$$

while the Reynolds stresses are specified directly at the inlet for the RSM.
Third-order MUSCL schemes are applied for the momentum and turbulence solvers with a coupled pressure-velocity scheme. A convergence tolerance of $1 \times 10^{-10}$ is set for all simulations to ensure the iterative errors are much smaller than the grid errors such that the former can be neglected.

5.5 Solution Verification

The results from the solution verification study for the RANS model simulations are shown in Table 4. The solution verification is performed only with the k-ω model. The distance metric to the asymptotic range is higher for $Re_\theta$ than $C_f$. Table 4 shows that monotonic convergence is achieved. The grid uncertainty is below 1.5% for both variables. The solution verification study shows that the bypass transition results are independent of the grid resolution.

Table 4: Solution verification for transitional boundary layer flow

<table>
<thead>
<tr>
<th></th>
<th>$Re_\theta$</th>
<th>$C_f$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R_G$</td>
<td>0.7298</td>
<td>0.7937</td>
</tr>
<tr>
<td>$p_G$</td>
<td>0.9087</td>
<td>0.6665</td>
</tr>
<tr>
<td>$P$</td>
<td>0.3029</td>
<td>0.2222</td>
</tr>
<tr>
<td>$U_G(%S_1)$</td>
<td>1.0583</td>
<td>0.2541</td>
</tr>
</tbody>
</table>

5.6 Results

The bypass transition simulation results are compared to the DNS results from Nolan and Zaki [1] and the CFD results by Ghasemi et al. [5]. The current simulations employ different inlet conditions and a finer mesh than the simulations by Ghasemi et al. The k, ω, ε, or Reynolds stress values are prescribed at the inlet, depending on the model in use, to match the conditions of the DNS simulation. The Ghasemi et al. inlet boundary conditions apply a
3% turbulent intensity. Additionally, this study examines entropy generation and compares CFD predictions with that post-processed from the DNS results.

Figure 13 shows how $R_e \theta$ varies with $R_e x^{1/2}$. The DNS data is linear in the log-log scale before $R_e x^{1/2} \approx 450$, where a slight bend occurs. The k-$\omega$ 4 equation model results closely resemble the DNS results but remain slightly lower throughout the domain. The slope in the log-log scale of the k-$\epsilon$ model remains linear throughout the entire domain, only showing similarity to the DNS results near the inlet. The k-$\epsilon$ model and RSM deviate from the DNS results before the k-$\omega$ and k-$\omega$ 4 equation models. The k-$\omega$ model shows close agreement to the DNS before $R_e x^{1/2} \approx 275$. The results from this study agree better with the DNS results than the results from Ghasemi et al.
Figure 14 and Figure 15 show how $C_t$ and $C_d$ vary with $Re_{x}^{1/2}$. The DNS data has a linear slope in the log-log scale until $Re_{x}^{1/2} \approx 450$ where both $C_t$ and $C_d$ increase sharply and level off again to a linear slope. The laminar region is the initial downward slope, the rise is transition region, and the small oscillations downstream are the fully turbulent region.

Similar to Figure 13, the $k$-$\omega$ 4 equation model agrees the best with the DNS data. The $C_d$ and $C_t$ predicted by the $k$-$\omega$ 4 equation model is 4% lower than the DNS data when $Re_{x}^{1/2} \approx 450$. The $k$-$\varepsilon$ model and RSM become transitional at the inlet and remain turbulent throughout the flow field. The $k$-$\omega$ model shows a laminar region until $Re_{x}^{1/2} \approx 200$, when transition onset occurs. The $k$-$\omega$ model shows close agreement to the DNS data in the turbulent region but transition occurs before the $k$-$\omega$ 4 equation model and the DNS data.

The results from this study agree better with the DNS results than the results from Ghasemi et al.
Figure 14: $C_t$ versus $Re_x^{1/2}$
Figure 15: $C_d$ versus $Re_x^{1/2}$

Figure 16 shows that all turbulence models examined herein predict transition onset earlier than the DNS data. The k-ω 4 equation model is the closest to the DNS data in predicting the transition onset location but over-predicts $\gamma$ by as much as 5% in the fully turbulent region when compared to the DNS results. The k-ω, k-ε, and RSM’s demonstrate very similar trends with steeper slopes than the other models.
5.7 Conclusions

Overall the present study shows significant improvements over the CFD results by Ghasemi et al. due to the employment of a finer grid and more accurate inlet boundary conditions for turbulent structures. The results show that the $k-\omega$ 4 equation model accurately predicts the boundary layer behavior and entropy generation for bypass-transitional flows. All models predict transition onset upstream of the location shown by the DNS data except the $k-\omega$ 4 equation model. This model shows a 4% difference in the onset of transition from the DNS data based on the skin friction coefficient.
Bypass transitional flow involves unsteady, freestream fluctuations to induce transition. Unfortunately, the time averaging of the RANS model would eliminate any benefit gained from running an unsteady simulation.
Chapter 6: Bypass Transition Boundary Layer Flows Using Large Eddy Simulation and Improved Delayed Detached Eddy Simulation

6.1 Objective and Approach

The IDDES and LES models are popular methods for the numerical solution of turbulent and transitional flow problems. LES only resolves the dynamically important scales, making it far less computationally expensive and time consuming than DNS. Application of DNS to turbomachinery flows is often too expensive, making LES a more feasible option. Correctly modeling inflow turbulence and properly resolving the flow physics are problems that must be solved with any LES to achieve physically realistic results.

While DNS resolves the entire spectrum of turbulent scales, LES only resolves the large eddies and models the small eddies. This technique can be applied because the smaller eddies do not transport as much momentum, mass, or energy and are less dependent on the geometry. LES splits the solution into two parts: the resolved zone and the modeled SGS zone. The variation of the SGS model can be categorized into either implicit or explicit formulations of the NS equations. The large scale motions of the flow are solved for by filtering out the small and universal eddies. For implicit filtering, LES resolves length scales based on the size of the grid spacing. For explicit filtering, one can specify the filter width.

In this study, preliminary results from the application of the Smagorinsky-Lilly dynamic model [50-52] and the IDDES model [53-55] are shown. The wall adapted large eddy simulation (WALES) model [56] is also tested with results not shown here.
6.2 Filtering

The difference between the LES model and DNS is filtering. Filtering occurs inside
the given SGS model. The implicit LES modeling relies on the numerical errors of the
particular spatial and temporal discretization schemes. The explicit LES modeling separates
the numerical and modeling errors by changing the filter width and the grid size. Some
models are considered to have both implicit and explicit filtering. One such model is the
dynamic model developed by Germano et al. [52] which applies an explicit filtering step to
compute the SGS stress tensor to the Smagorinsky SGS model.

6.3 Governing Equations

The LES equations are the time dependent NS equations that are filtered in either the
Fourier space or configuration space to eliminate eddies with scales smaller than the grid
spacing. The incompressible, filtered NS equations are,

\[ \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho \ddot{u}_i) = 0 \]  
\[ \frac{\partial}{\partial t} (\rho \ddot{u}_i) + \frac{\partial}{\partial x_j} (\rho \ddot{u}_i \ddot{u}_j) = \frac{\partial}{\partial x_j} (\sigma_{ij}) - \frac{\partial p}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} \]  

where \( \sigma_{ij} \) is the stress tensor due to molecular viscosity defined as,

\[ \sigma_{ij} = \left[ \mu \left( \frac{\partial \ddot{u}_i}{\partial x_j} + \frac{\partial \ddot{u}_j}{\partial x_i} \right) \right] - \frac{2}{3} \mu \frac{\partial \ddot{u}_i}{\partial x_i} \delta_{ij} \]  

6.3.1 Smagorinsky-Lilly Dynamic LES Model

The sub-grid scale eddy viscosity for the Smagorinsky model is,
\[ \mu_t = \rho L_s^2 |S_T| \]  

where \( L_s \) is the mixing length for the SGS and is defined in FLUENT as,

\[ L_s = \min\left(0.41y, C_s V^{1/3}\right) \]  

where \( V \) is the volume of the computational cell and \( C_s \) is the Smagorinsky constant. The dynamic Smagorinsky-Lilly model replaces the constant \( C_s \) with the following equations,

\[ L_{ij} = \left( \frac{\rho u_i' u_j'}{\rho u_i' u_i'} \right) - \frac{1}{\rho} \left( \frac{\rho u_i' u_j'}{\rho u_i' u_i'} \right) \]

\[ M_{ij} = -2 \left( (2\Delta)^2 \rho S_T S_{ij} - \Delta^2 \rho S_T S_{ij} \right) \]  

\[ C_s^2 = \frac{(L_{ij} - L_{kk} \delta_{ij})/3}{M_{ij} M_{ij}} \]  

where \( \Delta \) is the grid filter width. The SGS turbulent stresses are computed as,

\[ \tau_{ij} - \frac{1}{3} \tau_{kk} \delta_{ij} = -\rho v' \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \]  

6.3.2 IDDES Model

The IDDES model treats the near wall region in a RANS-like manner and the rest of the flow in an LES-like manner [57]. This approach avoids the need for a DNS grid resolution in the near wall region like LES models require. The IDDES model is based on the SST model in the near wall region and the LES model in the freestream region. Modifications are made to the SST model in the form of an additional sink term in the TKE transport Eq. (24),
\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right) + G_k - \frac{\rho \sqrt{k^3}}{l_{IDDES}}
\]  

(57)

where \(l_{IDDES}\) is a length scale based on the RANS turbulent length scale and the LES grid length scale. The equation for the length scale is complicated and will only be briefly discussed here. The general IDDES formulation from Spalart et al. [58] is,

\[
l_{IDDES} = l_{RANS} - \left(1 - \tanh\left(8r_d\right)^3\right)\max\left\{0, (l_{RANS} - l_{LES})\right\}
\]  

(58)

### 6.4 Post-Processing

Unlike the previously mentioned RANS model study, the LES and IDDES models require both three-dimensional and unsteady simulations, necessitating different post-processing methodologies. The total TKE from the simulations is the sum of the resolved and modeled TKE [59]. The resolved TKE is evaluated using the resolved velocity fluctuations. Reynolds decomposition dictates that the instantaneous velocity component is split into the mean and fluctuating components.

\[
u_i\{x, y, z, t\} = \bar{\bar{u}}_i\{x, y, z\} + u'_i\{x, y, z, t\}
\]  

(59)

The instantaneous and mean velocity components are calculated by FLUENT. The time-averaged velocity component is evaluated as,

\[
\bar{\bar{u}}_i\{x, y, z, t\} = \frac{1}{T_0} \int_0^{t+T_0} u\{x, y, z, t\} \, dt
\]  

(60)

where \(T_0\) is the averaging time. The resolved Reynolds stresses are calculated as,
and the modeled Reynolds stresses are calculated using Eq. (23). At each time step, the velocity fluctuations squared are calculated with Eq. (59) as well as the sum of the velocity fluctuations squared, 

\[ q^2 = (u'^2 + v'^2 + w'^2) \]  

These variables are averaged over time. The resolved Reynolds stresses are added to the corresponding modeled Reynolds stresses for the total Reynolds stresses in the entropy generation calculations, Eq. (42).

### 6.5 Simulation Setup

The inlet and outlet boundary conditions are similar to the boundaries for the RANS models. The spectral synthesizer within FLUENT is creates the inlet turbulence with profiles of the variables \( u, v, w, k, \epsilon, \bar{u}'u', \bar{v}'v', \bar{w}'w', \bar{u}'v', \bar{v}'w', \) and \( \bar{u}'w' \) specified at the inlet. The left and right span-wise boundary conditions are periodic. A second-order, implicit unsteady solver, central difference momentum solver, and second-order pressure solver are implemented. The SIMPLE scheme is implemented for the pressure-velocity coupling. A second-order implicit transient formulation is applied. The convergence tolerance within each time step is \( 1 \times 10^{-5} \). The ANSYS FLUENT User’s Manual details the steps to implementing the LES and IDDES models. First, a steady simulation is run to convergence using the transitional k-\( \omega \) 4 equation model on the three dimensional grid. The LES or IDDES model is then activated and the simulation is run until the flow is statistically steady.
Statistical convergence of running mean on the time history of the resistance establishes statistically stationary unsteady solutions [60]. Statistical convergence for the unsteady simulations is determined using the drag coefficient, $C_D$, defined as,

$$C_D = \int_{x_1}^{x_2} C_t \, dx$$ (63)

The drag coefficient is monitored during the simulation. Data is collected once the drag coefficient oscillations around a single value vary by only 1% or less. Once the flow is statistically steady, the flow statistics are sampled over time for post-processing. The time step for the simulations is $\Delta t^+ = 0.05$

6.6 Preliminary Results

The LES and IDDES results are compared with the results of RANS and DNS results. All variables plotted demonstrate large differences between the LES and IDDES models. If not explicitly specified, the variables are time-averaged. Figure 17 shows how $Re_\theta$ varies with $Re_x^{1/2}$. The LES data remains linear in the log-log scale up to $Re_x^{1/2} \approx 750$. The IDDES model shows early onset of transition, similar to the $k-\omega$ model, but diverges from the $k-\omega$ model around $Re_x^{1/2} \approx 400$. The IDDES model agrees well with the DNS data again around $Re_x^{1/2} = 600$. 

Figure 17: $Re_\theta$ versus $Re_x^{1/2}$

Figure 18 and Figure 19 show the IDDES and LES have similar trends between the $C_f$ and $C_d$. The LES model shows a linear function in the log-log scale for the entirety of the plate. The IDDES initially increases around $Re_x^{1/2} = 200$ similar to the $k-\omega$ model. Before the IDDES model reaches a fully turbulent trend, it decreases back toward a laminar trend. Around $Re_x^{1/2} = 600$, the IDDES model sharply rises into a fully turbulent trend for the remainder of the plate.
Figure 18: $C_f$ versus $Re_x^{1/2}$
Figure 20 shows the intermittency for the models. Similar to the trends of the other variables, LES shows a laminar trend throughout the length of the plate. The IDDES model initially increases at $\eta_l = 0$, following the trend seen from the k-ω model and the RSM. The IDDES model then diverges from the two models and drops down close to the laminar trend of the LES only to sharply increase at $\eta_l = 0.5$ to a turbulent trend.
Figure 20: $\gamma$ versus $\eta_f$

Figure 21 shows the instantaneous contours of the DES shielding function across the length of the plate. The lower the number, the more the field is solved using the RANS model while the higher the number, the more the field is solved using the LES model.

Figure 21: DES Shielding Function Contour

Figure 22 and Figure 23 show instantaneous contours of $\bar{u}'\bar{v}'$ at two different $x$ locations. The first $x$ location is very close to the inlet and shows the boundary layer is
uniform in the spanwise direction and the inlet freestream turbulence is observed. The second x location is further downstream and significant turbulence within the boundary layer is observed.

Figure 22: IDDES contour of $\bar{u}'\bar{v}'$ at $x/L_x = 0.04$
6.7 Conclusions

Bypass transition flow is modeled using the LES and IDDES models. The entropy generation, skin friction coefficient, intermittency, and Reynolds number based on momentum thickness is calculated from the time averaged results of these simulations. These results are compared to the results from similar RANS and DNS simulations of bypass transition.

The LES model did not predict the onset of transition. This is possibly caused by several factors. One factor could be the grid spacing in the streamwise direction is too large. As pointed out in Ferziger [39], the maximum wavenumber, \( k_{\text{max}} \), that can be represented is \( k_{\text{max}} = \pi/2\Delta x \). The bulk of the energy should be contained in the wavenumbers inside this
limit. If the bulk of the energy is not within this limit, the energy is lost and the turbulence is dissipated. Due to computational costs and limitations of the computational resources, a finer grid in the streamwise direction was not feasible, possibly allowing the freestream turbulence to be dissipated. The freestream turbulence causes the onset of bypass transition, therefore, dissipation of the freestream turbulence would prevent the onset of transition. The second factor could be that the orders of accuracy for the numerical schemes need to be improved. The discretization of the numerical methods could be even larger than the contributions from the LES SGS models for the currently applied second-order scheme.

Because the SST k-ω model is used in the IDDES model, the first onset of transition correlates with the onset of transition seen with the two-dimensional, steady SST k-ω model. The first onset of transition descends back toward a laminar trend then sharply ascends into a fully turbulent trend. This is clearly seen in Figure 18 and Figure 19. It is not explicitly known why the initial transition descends back toward a laminar trend and ascends to a turbulent trend further downstream. It is hypothesized that the damping of the freestream turbulence is due to a large streamwise grid size. This damping of the freestream turbulence impacts the boundary layer transition, causing the initial descent of the $C_t$ and $C_d$ values. The second ascent is hypothetically caused by a decrease in the freestream turbulence maximum wavenumber. This decrease in the maximum wavenumber prevents the streamwise grid size from dissipating the freestream turbulence. This turbulence induces transition within the boundary layer.
Chapter 7: Future Work

In the future, the RANS and IDDES models will be conducted for bypass transitional flow with streamwise pressure gradients. Sensitivity of the IDDES model grid resolution and time step size will be examined, which would provide hints to explain the noted trends of the $C_d$ variable in the streamwise direction. Attempts have been made to recreate the Orr-Sommerfeld hydrodynamic instabilities at the inlet within ANSYS FLUENT, which may lead to more accurate CFD predictions for bypass transitional boundary layer flows.
References


Appendices

Appendix A. Hydrodynamic stability

Hydrodynamic stability is a complicated subject. The Orr-Sommerfeld equations are used to determine stability in bypass transitional flows. The equations provided by Grosch and Halwen [61] are,

\[ \left( \frac{\partial^2}{\partial y^2} - \alpha^2 \right)^2 \phi = i\alpha Re_L \left[ \left( U_m - c \right) \left( \frac{\partial^2}{\partial y^2} - \alpha^2 \right) - \left( \frac{\partial^2 U_m}{\partial y^2} \right) \right] \phi \]  

(A1)

where all variables are dimensionless; the length scale is \( L \), the velocity scale is \( U_0 \), the wavenumber is \( \alpha \), and \( \phi \) is defined from the stream function, \( \psi \), as,

\[ \psi \{ x, y, t \} = \phi \{ y \} e^{i\alpha(x-c)} \]  

(A2)

The solution to the Orr-Sommerfeld equations can be obtained with a backward Runge-Kutta method, as outlined in Schmid and Henningson, 2001 [62]. Due to the complexity of the Orr-Sommerfeld equations, a more detailed description is provided Jacobs and Durbin, 1998 [63]. Computation of a temporal eigenfunction is shown in Figure 24. The conditions \( \alpha = 0.179, Re_L = 580 \), and \( k_y = 0.358 \) as shown in Figure 1 in Jacobs and Durbin [63]. Multiple codes are in place to calculate the eigenfunction. These codes are listed in the code section of the appendix.
Figure 24: Temporal eigenfunction
Appendix B. Code

B.1 Orr-Sommerfeld Solution (Written in Matlab)

clear
clc
global R alp h U fdd fddd y ns phi;
R = 580;
alp = 0.179;
h = 0.005;
ky = 2;
c = 1 - 1i*(1 + ky^2)*alp/R;
fend = 50;
[U,fdd,fddd,y] = Blasiusdimless(0,50/sqrt(2),h/(2*sqrt(2)));
N = length(y);
ns = fend/h;
N = length(y);
if N ~= 2*ns
    disp('Error with matrix sizes between Blasius solution')
end
lambda1 = -1i*ky*alp;
lambda2 = 1i*ky*alp;
lambda3 = -alp;
for j = 1:4
    phi1t = lambda1^(j-1)*exp(-1i*ky*alp*(fend - h));
    phi2t = lambda2^(j-1)*exp(1i*ky*alp*(fend - h));
    phi3t = lambda3^(j-1)*exp(-alp*(fend - h));
    phi(j,:) = [phi1t phi2t phi3t];
end
[psi1, psi2, psi3, psi4] = Back_RungeKutta_OS(c);
psi1(ns,:) = phi(1,:);
AM = [psi1(1,2) psi1(1,3);
     psi2(1,2) psi2(1,3)];
b = [psi1(1,1);
     psi2(1,1)];
[coeff] = GaussElimination(AM,-b,2);% [coeff] = linsolve(AM,b);
coeff = [1; coeff(1); coeff(2)];
for j = 1:ns
    psifinal(j) = psi1(j,:)*coeff;
y1(j) = y(2*j - 1);
end
d = [y1',real(psifinal)',imag(psifinal)']
% xlswrite('psifinal.xls',d);
plot(y1,real(psifinal))
hold on
plot(y1,imag(psifinal))
hold off
function [ bstar ] = GaussElimination( A,b,n )
% A is the matrix being brought it, b is the vector of the solutions
% and n is the size of the solution
Aug = [A,b];
for j = 1:n
    while Aug(j,j) == 0
        temp = Aug(j,:);
        Aug(j,:) = Aug(j+1,:);
        Aug(j+1,:) = temp;
        Aug(j,j) = -Aug(j,j);
    end
    Aug(j,:) = Aug(j,:)/Aug(j,j);
    for k = (j + 1):n
        Aug(k,:) = -Aug(k,j)*Aug(j,:) + Aug(k,:);
    end
    if j == 1
        continue
    else
        for k = 1:j-1
            Aug(k,:) = -Aug(k,j)*Aug(j,:) + Aug(k,:);
        end
    end
end
if size(b,2)> 1
    for j = 1:n
        bstar(:,j) = Aug(:,j + n);
    end
else
    bstar = Aug(:,n + 1);
end
end

function [ psi1new, psi2new, psi3new, psi4new ] = Back_RungeKutta_OS( c )
% This solves the spectrum for phi using the OS equation for a given
% dimensionless velocity profile from the blasius solution (U) and its
% second derivative(fdd), an initial condition to phi (phi), a constant
% alpha and reynolds number (alp and R respectively), an initial value for
% c
% (c), and a set # of iterations (ns) which must be half the number of
% iterations used in the blasius solution
% % [ lambdanew ]...
% % = RungeKutta_OS_eigenvalues( c )
global R alp h U fdd fddd ns phi;
for j = ns:-1:2
    k = 2*j;
    Uf = U(k);
    fcurrent = fddd(k);
    a = -1i*alp*R*(alp^2*(Uf - c) + fcurrent) - alp^4;
    b = 1i*alp*R*(Uf - c) + 2*alp^2;
    A1 = [0 1 0 0;
          0 0 1 0;]
\[
\begin{align*}
0 & 0 & 0 & 1; \\
a & 0 & b & 0];
\end{align*}
\]

\[
\begin{align*}
U_f &= U(k - 1); \\
f_{current} &= f_{dd}(k - 1); \\
a &= -1i*\alpha*R*(\alpha^2*(U_f - c) + f_{current}) - \alpha^4; \\
b &= 1i*\alpha*R*(U_f - c) + 2*\alpha^2; \\
A_2 &= [0 1 0 0; \\
0 0 1 0; \\
a 0 b 0];
\end{align*}
\]

\[
\begin{align*}
U_f &= U(k - 2); \\
f_{current} &= f_{dd}(k - 2); \\
a &= -1i*\alpha*R*(\alpha^2*(U_f - c) + f_{current}) - \alpha^4; \\
b &= 1i*\alpha*R*(U_f - c) + 2*\alpha^2; \\
A_3 &= [0 1 0 0; \\
0 0 1 0; \\
a 0 b 0];
\end{align*}
\]

\[
\begin{align*}
g_1 &= h*A_1*\phi; \\
g_2 &= h*A_2*(\phi + g_1/2); \\
g_3 &= h*A_2*(\phi + g_2/2); \\
g_4 &= h*A_3*(\phi + g_3); \\
psi_{1\text{new}}(j - 1,:) &= \phi(1,:) - (g_1(1,:) + 2*g_2(1,:) + 2*g_3(1,:) + \\
g_4(1,:))/6; \\
psi_{2\text{new}}(j - 1,:) &= \phi(2,:) - (g_1(2,:) + 2*g_2(2,:) + 2*g_3(2,:) + \\
g_4(2,:))/6; \\
psi_{3\text{new}}(j - 1,:) &= \phi(3,:) - (g_1(3,:) + 2*g_2(3,:) + 2*g_3(3,:) + \\
g_4(3,:))/6; \\
psi_{4\text{new}}(j - 1,:) &= \phi(4,:) - (g_1(4,:) + 2*g_2(4,:) + 2*g_3(4,:) + \\
g_4(4,:))/6;
\end{align*}
\]

\[
\begin{align*}
\phi &= [\psi_{1\text{new}}(j - 1,:); \\
\psi_{2\text{new}}(j - 1,:); \\
\psi_{3\text{new}}(j - 1,:); \\
\psi_{4\text{new}}(j - 1,:)];
\end{align*}
\]

end

end

function [ U, f2d, ftd, y ] = Blasiusdimless( beta, finalpoint, deta )
% Solves the Blasius solution for f', f'' and eta. Note that deta is with
% respect to the change in eta which is sqrt(2)*y
% [ U, y ] = Blasiusdimless( beta, finalpoint, deta )
fd(1,1) = 0;
f(1,1) = 0;
fddd(1,1) = 0;
eta(1,1) = 0;
ns = finalpoint/deta;
deta = deta;
finalpoint = finalpoint;
stopping_distance = 0.0000000001;
shoot_parameter(1,1) = 1;
fdd(1,1) = shoot_parameter(1,1);

for i = 1:ns
    eta(i,1) = delta*(i-1);
    fp = f(i,1) + delta*fd(i,1);
    fdp = fd(i,1) + delta*fdd(i,1);
    fddp = fdd(i,1) + delta*fddd(i,1);
    fdddp = -fp*fdp - beta*(1 - fdp^2);
    f(i + 1,1) = f(i,1) + delta*(fd(i,1) + fdp)/2;
    fd(i + 1,1) = fd(i,1) + delta*(fdd(i,1) + fddp)/2;
    fdd(i + 1,1) = fdd(i,1) + delta*(fddd(i,1) + fdddp)/2;
    fddd(i + 1,1) = -f(i + 1,1)*fdd(i + 1,1) - beta*(1 - fd(i + 1,1)^2);
end
f_dash_endpoint(1,1) = fd(ns,1);

shoot_parameter(2,1) = 1.1;
n = 2;
while abs(shoot_parameter(n,1) - shoot_parameter(n-1,1)) > stopping_distance
    fdd(1,1) = shoot_parameter(n,1);
    fd(1,1) = 0;
    f(1,1) = 0;
    fddd(1,1) = 0;
    for i = 1:ns
        eta(i,1) = delta*(i-1);
        fp = f(i,1) + delta*fd(i,1);
        fdp = fd(i,1) + delta*fdd(i,1);
        fddp = fdd(i,1) + delta*fddd(i,1);
        fdddp = -fp*fdp - beta*(1 - fdp^2);
        f(i + 1,1) = f(i,1) + delta*(fd(i,1) + fdp)/2;
        fd(i + 1,1) = fd(i,1) + delta*(fdd(i,1) + fddp)/2;
        fdd(i + 1,1) = fdd(i,1) + delta*(fddd(i,1) + fdddp)/2;
        fddd(i + 1,1) = -f(i + 1,1)*fdd(i + 1,1) - beta*(1 - fd(i + 1,1)^2);
    end
    f_dash_endpoint(n,1) = fd(ns,1);
    shoot_parameter(n+1,1) = shoot_parameter(n,1) - (f_dash_endpoint(n,1) - 1)*(shoot_parameter(n,1) - shoot_parameter(n-1,1))/(f_dash_endpoint(n,1) - f_dash_endpoint(n-1,1));
    n = n + 1;
end
for i = 1:ns
    y(i,1) = eta(i,1)*sqrt(2);
    U(i,1) = fd(i,1);
    f2d(i,1) = fdd(i,1)*sqrt(2);
    ftd(i,1) = fddd(i,1)*sqrt(2);
end
B.2 Post-processing (Written in SciLab)

clear
clear
clc
% All data that will be referenced
global Uinfperc rho mu model nu dt
dt = date()
% The percentage of the freestream velocity for which the boundary layer is determined
Uinfperc = 0.99
% Density
rho = 1.225
% Kinematic Viscosity
mu = 0.00004
nu = mu/rho
% Model will be the corresponding RANS, LES, or DES model
model = 'LES'

clear
clc
global rho mu nu Uinfperc model dt
exec DDX.sci;

filn = 'InputFiles/'+model+'_Test/'
% Imports the .txt files that are provided by FLUENT
xun1 = fscanfMat(filn+'xcoor1.txt')
yun1 = fscanfMat(filn+'ycoor1.txt')
xvelun1 = fscanfMat(filn+'xvel1.txt')
yvelun1 = fscanfMat(filn+'yvel1.txt')
tauun1 = fscanfMat(filn+'tau1.txt')
cfun1 = fscanfMat(filn+'cf1.txt')
xun2 = fscanfMat(filn+'xcoor2.txt')
yun2 = fscanfMat(filn+'ycoor2.txt')
xvelun2 = fscanfMat(filn+'xvel2.txt')
yvelun2 = fscanfMat(filn+'yvel2.txt')
tauun2 = fscanfMat(filn+'tau2.txt')
cfun2 = fscanfMat(filn+'cf2.txt')
uvun1 = fscanfMat(filn+'uvresolved1.txt')
uvun2 = fscanfMat(filn+'uvresolved2.txt')
vwun1 = fscanfMat(filn+'vwresolved1.txt')
vwun2 = fscanfMat(filn+'vwresolved2.txt')
uwun1 = fscanfMat(filn+'uwresolved1.txt')
uwun2 = fscanfMat(filn+'uwresolved2.txt')
turbnuun1 = fscanfMat(filn+'turbnu1.txt')
turbnuun2 = fscanfMat(filn+'turbnu2.txt')
if model == 'DES' then
    tkeun1 = fscanfMat(filn+'tke1.txt')
    tkeun2 = fscanfMat(filn+'tke2.txt')
    tkeun = [tkeun1;tkeun2]
end
uuun1 = fscanfMat(filn+'uumean1.txt')
uuun2 = fscanfMat(filn+'uumean2.txt')
vvun1 = fscanfMat(filn+'vvmean1.txt')
vvun2 = fscanfMat(filn+'vvmean2.txt')
qun1 = fscanfMat(filn+'q1.txt')
qun2 = fscanfMat(filn+'q2.txt')
dudxun1 = fscanfMat(filn+'dudx1.txt')
dudxun2 = fscanfMat(filn+'dudx2.txt')
dudyun1 = fscanfMat(filn+'dudy1.txt')
dudyun2 = fscanfMat(filn+'dudy2.txt')
dvxun1 = fscanfMat(filn+'dvdx1.txt')
dvxun2 = fscanfMat(filn+'dvdx2.txt')
xun = [xun1;xun2]
yun = [yun1;yun2]
xvelun = [xvelun1;xvelun2]
yvelun = [yvelun1;yvelun2]
tauun = [tauun1;tauun2]
%strainrun = [strainrun1;strainrun2]
cfun = [cfun1;cfun2]
uvun = [uvun1;uvun2]
vwun = [vwun1;vwun2]
uwun = [uwun1;uwun2]
turbnuun = [turbnuun1;turbnuun2]
qun = [qun1;qun2]
uuun = [uuun1;uuun2]
vvun = [vvun1;vvun2]
dudyun = [dudyun1;dudyun2]
dvxun = [dvxun1;dvxun2]
dvxun = [dvxun1;dvxun2]

[xmod,kc] = gsort(xun(:,2),'r','i')
nrx = size(xmod,1)
k = 0
rk = 1
kkj = 1
ml = spget(sparse(tauun(:,2)))
for j = 1:nrx
    kj = kc(j,1)
    if cfun(kj,2) > 0 then
        kk = kk + 1
        x(kk,1) = xmod(j,1)
        tkj = ml(kk,1)
        tau(kk,1) = tauun(tkj,2)
        Uttau(kk,1) = sqrt(tau(kk,1)/rho)
\[
\text{rowl}(kk,1) = rk-1 \\
\text{rk} = 1 \\
\text{end}
\]
\[
\text{yn}(rk,kk) = yun(kj,2) \\
\text{xvel}(rk,kk) = xvelun(kj,2) \\
\text{yvel}(rk,kk) = yvelun(kj,2) \\
\%\text{strain}(rk,kk) = strainrun(kj,2) \\
\text{if model == 'DES' then} \\
\quad \text{tk}(rk,kk) = tkeun(kj,2) \\
\text{end}
\]
\[
\text{uvm}(rk,kk) = uvun(kj,2) \\
\text{vwm}(rk,kk) = vwun(kj,2) \\
\text{uwm}(rk,kk) = uwun(kj,2) \\
\text{qm}(rk,kk) = qun(kj,2) \\
\text{uum}(rk,kk) = uuun(kj,2) \\
\text{vvm}(rk,kk) = vvun(kj,2) \\
\text{dudxm}(rk,kk) = dudxun(kj,2) \\
\text{dvdxm}(rk,kk) = dvdxun(kj,2) \\
\text{dudym}(rk,kk) = dyun(kj,2) \\
\text{turbnum}(rk,kk) = turbnuun(kj,2) \\
\text{rk} = rk + 1 \\
\text{end}
\]
\[
\text{rowl}(l,1) = [] \\
\text{rowl}(kk,1) = rk-1 \\
\text{nrx} = \text{size}(x,1) \\
\%\text{The following removes repeat y points}
\text{for j = 1:nrx}
\quad [y(:,j),kcv] = \text{unique}(ym(:,j), 'r') \\
\quad kcv(1,1) = 1 \\
\quad xvel(:,j) = xvelm(kcv,j) \\
\quad yvel(:,j) = yvelm(kcv,j) \\
\quad %\text{strain}(:,j) = strainm(kcv,j) \\
\quad uv(:,j) = uvm(kcv,j) \\
\quad vw(:,j) = vwm(kcv,j) \\
\quad uw(:,j) = uwm(kcv,j) \\
\quad q(:,j) = qm(kcv,j) \\
\text{if model == 'DES' then} \\
\quad tke(:,j) = tkem(kcv,j) \\
\text{end}
\quad uu(:,j) = uum(kcv,j) \\
\quad vv(:,j) = vvm(kcv,j) \\
\quad dudx(:,j) = dudxm(kcv,j) \\
\quad dvdx(:,j) = dvdxm(kcv,j) \\
\quad dudy(:,j) = dyun(kcv,j) \\
\quad yplus(:,j) = y(:,j).*Utau(j,1)/nu \\
\quad turbnu(:,j) = turbnum(kcv,j)/rho \\
\quad kl(j,1) = \text{size}(y(:,j),1)
\text{end}
\text{for j = 1:nrx}
\quad \text{for k = 1:kl(j,1)}
\quad \quad uvt(k,j) = uv(k,j) - turbnu(k,j)*rho*(dudy(k,j) + dvdx(k,j)) \\
\text{if model == 'DES' then}
\[ uut(k, j) = uu(k, j) + \frac{2}{3}tke(k, j) - 2\cdot\text{turbnu}(k, j)\cdot\rho\cdot(dudx(k, j)) \]

\[ \text{elseif model == 'LES'} \]
\[ uut(k, j) = uu(k, j) - 2\cdot\text{turbnu}(k, j)\cdot\rho\cdot(dudx(k, j)) \]

\[ \text{end} \]

\[ \text{end} \]

\[ kk = 1 \]
\[ \text{while } xvel(kk, j) < \text{Uinfperc*max(xvel(:, j))} \]
\[ kk = kk + 1 \]
\[ \text{end} \]

\[ \text{Uinf}(j, 1) = xvel(kk, j) \]
\[ \text{cf}(j, 1) = \frac{\tau(j, 1)}{0.5\cdot\rho\cdot\text{Uinf}(j, 1)} \]
\[ \text{delta}(j, 1) = y(kk, j) \]
\[ \text{Rex}(j, 1) = \frac{\text{Uinf}(j, 1)\cdot x(j, 1)}{\nu} \]
\[ \text{sqrtRex}(j, 1) = \sqrt{\text{Rex}(j, 1)} \]
\[ \text{kv}(j, 1) = kk \]

\[ \text{end} \]

\[ \text{uv} = uvt \]

\[ \text{uu} = uut \]

\[
\text{mkdir('OutputFiles', model+'test'+dt)}
\text{mkdir('ResultFiles', model+'test'+dt)}
\text{filn = 'OutputFiles/'+model+'test'+dt}
\text{filnres = 'ResultFiles/'+model+'test'+dt}
\text{save(filn+val.sod',x,y,yplus,cf,xvel,yvel,Utau,tau,kl,kv,Uinf,sqrtRex...}
\text{,Rex, delta, dudy, dvdx, dudx, uv, vw, uw, q, uu, vv, turbnu)}
\]

\[ \text{clear} \]

\[ \text{clc} \]

\[ \text{global rho mu nu model dt} \]

\[ \text{exec Integration.sci;} \]
\[ \text{exec DDX.sci;} \]

\[ \text{filn = 'OutputFiles'+model+'test'+dt} \]
\[ \text{load(filn+val.sod')} \]
\[ \text{nrx = size(x,1)} \]
\[ \text{ncy = size(y,2)} \]
\[ \text{nry = size(y,1)} \]

\[ \text{if ncy ~= nrx then} \]
\[ \text{disp('Error with size of matrices')} \]
\[ \text{disp(['X size',mtlb_num2str(nrx)])} \]
\[ \text{disp(['Y size',mtlb_num2str(ncy)])} \]
\[ \text{end} \]

\[ \text{for j = 1:nrx} \]
\[ \text{for k = 1:1:kl(j,1)} \]
\[ \text{funct(k,j) = (1 - xvel(k,j)/Uinf(j,1))*xvel(k,j)/Uinf(j,1)} \]
\[ \text{end} \]

end

[mom] = Integration(y,funct,kv)

for j = 1:nrx
    Reth(j,1) = mom(j,1)*Uinf(j,1)/nu
end

filnres = 'ResultFiles/'+model+'test'+dt

disp('Reth calculated')
%********************************************************************
%*************
%**********************************************
%Which output you would like to see
% 1: Pointwise entropy generation
% 2: Intermittency
% 3: All of the above
%**********************************************

out = 3

nu = mu/rho

nrx = size(x,1)

%********************************************************************
%********************************************************************
%************************************************************
if out == 1 | out == 3 then
    cdterm1 = Integration(y,mu.*(dudy.^2 + dvdx.^2),kl)
    disp('cdterm1 done')
    cdterm2 = Integration(y,rho.*uv.*dudy,kv)
    disp('cdterm2 done')
    cdterm3 = Integration(y,rho.*uu-vv).*dudx,kv)
    disp('cdterm3 done')
    entrt = real(Integration(y,0.5*rho.*xvel.*q,kv))
    cdterm4 = DDX(x,entrt)
    disp('cdterm4 done')
    for j = 1:nrx
        kkc = kl(j,1)
        Cd1(j,1) = cdterm1(j,1) - cdterm2(j,1) - cdterm3(j,1) - cdterm4(j,1)
    end
    for j = 1:nrx
        Cd(j,1) = (Cd1(j,1))/(rho*Uinf(j,1)^3)
    end
    disp('Cd calculated')
end
%************************************************************
%********************************************************************
%********************************************************************
if out == 2 | out == 3 then
    for j = 1:nrx
        Turbcf(j,1) = 0.455/((log(0.06*Rex(j,1)))^2)
        Lamcf(j,1) = 0.664/(sqrt(Rex(j,1)))
        intermit(j,1) = (cf(j,1) - Lamcf(j,1))/(Turbcf(j,1) - Lamcf(j,1))
    end
end
end
if intermit($,1) < 0.005 then
    trs = 1
else
    k = 1
    while intermit(k,1) < 0.005
        k = k + 1
    end
    trs = k
end
for j = 1:nrx
    eta(j,1) = (x(j,1) - x(trs,1))/(x($,1) - x(trs,1))
end
disp('Gamma calculated')
end
save(filnres+'\val.sod',Reth,Cd,eta,intermit,Turbcf,Lamcf)

function [ mom ] = Integration(coord,funct,kv)
    if size(coord,2) > 1 then
        nrx = size(coord,2)
        for j = 1:nrx
            theta = 0
            f1 = funct(1,j)
            f2 = funct(2,j)
            f3 = funct(3,j)
            fvec = [f1;f2;f3]
            y1 = coord(1,j)
            y2 = coord(2,j)
            y3 = coord(3,j)
            Am = [y1^2, y1, 1;
                   y2^2, y2, 1;
                   y3^2, y3, 1];
            [vff] = linsolve(Am,-fvec)
            CA = vff(1)
            CB = vff(2)
            CC = vff(3)
            theta = theta + CA/3*(y3^3-y1^3)+CB/2*(y3^2-y1^2) + CC*(y3-y1)
        for k = 3:2:kv(j,1)
            f1 = funct(k-1,j)
            f2 = funct(k,j)
            f3 = funct(k+1,j)
            fvec = [f1;f2;f3]
            y1 = coord(k-1,j)
            y2 = coord(k,j)
            y3 = coord(k+1,j)
            Am = [y1^2, y1, 1;
                   y2^2, y2, 1;
                   y3^2, y3, 1];
            [vff] = linsolve(Am,-fvec)
            CA = vff(1)
            CB = vff(2)
CC = vff(3)
theta = theta + CA/3*(y3^3-y1^3) + CB/2*(y3^2-y1^2) + CC*(y3-y1)
end
mom(j,1) = theta
end
else
nr = size(coord,1)
theta = 0
f1 = funct(1,1)
f2 = funct(2,1)
f3 = funct(3,1)
fvec = [f1,f2,f3]
y1 = coord(1,1)
y2 = coord(2,1)
y3 = coord(3,1)
Am = [y1^2, y1, 1;
y2^2, y2, 1;
y3^2, y3, 1];
[vff] = linsolve(Am,-fvec)
CA = vff(1)
CB = vff(2)
CC = vff(3)
theta = theta + CA/3*(y3^3-y1^3)+CB/2*(y3^2-y1^2) + CC*(y3-y1)
for k = 3:2:nrx-1
    f1 = funct(k-1,1)
f2 = funct(k,1)
f3 = funct(k+1,1)
fvec = [f1,f2,f3]
y1 = coord(k-1,1)
y2 = coord(k,1)
y3 = coord(k+1,1)
Am = [y1^2, y1, 1;
y2^2, y2, 1;
y3^2, y3, 1];
[vff] = linsolve(Am,-fvec)
CA = vff(1)
CB = vff(2)
CC = vff(3)
theta = theta + CA/3*(y3^3-y1^3)+CB/2*(y3^2-y1^2) + CC*(y3-y1)
end
f1 = funct(nrx-3,1)
f2 = funct(nrx-2,1)
f3 = funct(nrx-1,1)
fvec = [f1,f2,f3]
y1 = coord(nrx-3,1)
y2 = coord(nrx-2,1)
y3 = coord(nrx-1,1)
Am = [y1^2, y1, 1;
y2^2, y2, 1;
y3^2, y3, 1];
[vff] = linsolve(Am,-fvec)
CA = vff(1)
CB = vff(2)
CC = vff(3)
theta = theta + CA/3*(y3^3-y1^3)+CB/2*(y3^2-y1^2) + CC*(y3-y1)
mom(1,1) = theta
end
endfunction

function [ Mf ] = MatrixComp(M1,M2)
nry1 = size(M1,1)
nrx1 = size(M1,2)
nry2 = size(M2,1)
nrx2 = size(M2,2)
if nry1 < nry2 then
    M1f = M1
    for j = 1:(nry2-nry1)
        M1f(nry1+j,:) = M1(:,j)
    end
    Mf = [M1f,M2]
elseif nry1 > nry2 then
    M2f = M2
    for j = 1:(nry1-nry2)
        M2f(nry2+j,:) = M2(:,j)
    end
    Mf = [M1,M2f]
else
    Mf = [M1,M2]
end
endfunction

function [ dfdx ] = DDX(coord,funct)
[rz,cz] = find(coord(2:,:,== 0))
coord(rz,:) = []
nrx = size(coord,1)
y1 = coord(1,1)
y0 = coord(2,1)
y1 = coord(3,1)
un1 = funct(1,1)
u0 = funct(2,1)
u1 = funct(3,1)
h1 = y0 - y1
h2 = y1 - y0
a = -(2*h1 + h2)/(h1*(h1+h2))
b = (h1 + h2)/(h1*h2)
c = -(h1)/(h2*(h1+h2))
dfdx(1,1) = a*un1 + b*u0 + c*u1
for j = 2:nrx-1
    y1 = coord(j-1,1)
y0 = coord(j,1)
y1 = coord(j+1,1)
un1 = funct(j-1,1)
u0 = funct(j,1)
u1 = funct(j+1,1)
\[ h_1 = y_0 - y_{n1} \]
\[ h_2 = y_1 - y_0 \]
\[ a = -(2h_1 + h_2)/(h_1*(h_1+h_2)) \]
\[ b = (h_1 + h_2)/(h_1*h_2) \]
\[ c = -(h_1)/(h_2*(h_1+h_2)) \]
\[ dfdx(j,1) = a*u_{n1} + b*u_0 + c*u_1 \]

end

\[ y_{n1} = \text{coord}(nrx-2,1) \]
\[ y_0 = \text{coord}(nrx-1,1) \]
\[ y_1 = \text{coord}(nrx,1) \]
\[ u_{n1} = \text{funct}(nrx-2,1) \]
\[ u_0 = \text{funct}(nrx-1,1) \]
\[ u_1 = \text{funct}(nrx,1) \]
\[ h_1 = y_0 - y_{n1} \]
\[ h_2 = y_1 - y_0 \]
\[ a = -(2h_1 + h_2)/(h_1*(h_1+h_2)) \]
\[ b = (h_1 + h_2)/(h_1*h_2) \]
\[ c = -(h_1)/(h_2*(h_1+h_2)) \]
\[ dfdx(nrx,1) = a*u_{n1} + b*u_0 + c*u_1 \]

endfunction