Numerical Investigation of Second-Order Effects in a Supersonic Boundary-Layer

Timothy R. Membrino

Embry-Riddle Aeronautical University - Daytona Beach

Follow this and additional works at: http://commons.erau.edu/db-theses

Part of the Aerospace Engineering Commons

Scholarly Commons Citation

This thesis is brought to you for free and open access by Embry-Riddle Aeronautical University – Daytona Beach at ERAU Scholarly Commons. It has been accepted for inclusion in the Theses - Daytona Beach collection by an authorized administrator of ERAU Scholarly Commons. For more information, please contact commons@erau.edu.
NUMERICAL INVESTIGATION OF SECOND-ORDER EFFECTS IN A SUPersonic
BOUNDARY-LAYER

by

Timothy Richard Membrino

A Thesis Submitted to the
Office of Graduate Programs
in Partial Fulfillment of the Requirements for the Degree of
Master of Aerospace Engineering

Embry-Riddle Aeronautical University
Daytona Beach, Florida
April 1995
INFORMATION TO USERS

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleed-through, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.
NUMERICAL INVESTIGATION OF SECOND-ORDER EFFECTS IN A SUPERSONIC BOUNDARY-LAYER

by

Timothy Richard Membrino

This thesis was prepared under the direction of the candidate's thesis committee chairman, Dr. Jose Rodriguez, Department of Aerospace Engineering, and has been approved by the members of his thesis committee. It was submitted to the Office of Graduate Programs and was accepted in partial fulfillment of the requirements for the degree of Master of Aerospace Engineering.

THESIS COMMITTEE:

Dr. Jose Rodriguez
Chairman

Dr. George Emanuel
Member

Dr. Tej Gupta
Member

Department Chair, Aerospace Engineering

Dean of Faculty, Daytona Beach campus

April 18, 1995
ACKNOWLEDGMENTS

The author wishes to express special thanks to the Thesis Chairman, Dr. Jose Rodriguez without whose consistent support, understanding, patience and experience this effort would not have been possible.

The comments and assistance of the thesis committee members, Dr. George Emanuel and Dr. Tej Gupta is greatly appreciated.

Thanks also to Sean Peiffer for his continuous hospitality and assistance with the logistics of completing a thesis in absentia.

Finally, thanks to my friends and family who provided much needed encouragement and motivation when it seemed unlikely I would complete this effort.
ABSTRACT

Author: Timothy R. Membrino
Title: Numerical Investigation of Second-Order Effects in a Supersonic Boundary-Layer
Institution: Embry-Riddle Aeronautical University
Degree: Master of Aerospace Engineering
Year: 1995

Historically, the study of boundary-layer flows has centered on the analysis of the first-order boundary-layer equations and their application to physical flow problems. However, selected “real-world” boundary-layer flows exhibit significant second-order effects which are neglected by the first-order equations. Full Navier-Stokes solutions are often not merited or desired for these flows. Therefore, the second-order boundary-layer equations provide a compromise.

Few validating comparisons have been attempted between second-order boundary-layer theory and experimental or numerical solutions of compressible viscous flows. Experimental simulations to capture second-order effects are difficult since the desired effects are small and can exist simultaneously, resulting in a neutralizing effect.¹

This report documents the application of Computational Fluid Dynamics (CFD) to the solution of second-order boundary-layer effects. Development of an optimum computational grid is the primary problem encountered. The effort involves significant analysis of the influence of various grid designs on the computational resolution. The numerical experimentation is performed for the supersonic flow over a flat plate at zero angle of attack. The second-order effects are initiated by the introduction of a stagnation enthalpy gradient in the flowfield at the plate leading edge.
TABLE OF CONTENTS

ACKNOWLEDGMENTS................................................................................................................ iii

ABSTRACT................................................................................................................................. iv

TABLE OF CONTENTS ............................................................................................................... v

LIST OF FIGURES ...................................................................................................................... viii

LIST OF SYMBOLS .................................................................................................................... viii

INTRODUCTION ......................................................................................................................... 1

SECOND-ORDER BOUNDARY-LAYER THEORY ....................................................................... 4

PERTURBATION METHODS (MATCHED ASYMPTOTIC EXPANSIONS) .................................... 4

  Inner Region (Boundary-Layer Equations) ............................................................................ 5

  Outer Region (External Flow Equations) ................................................................................ 6

  Boundary Conditions ............................................................................................................ 7

  Inner Equations (First-Order) .............................................................................................. 7

  Inner Equations (Second-Order) .......................................................................................... 7

  Outer Equations (First-Order) ............................................................................................. 8

  Outer Equations (Second-Order) ........................................................................................ 8

  Interface Matching Conditions (Inner Variables) ............................................................... 9

SECOND-ORDER EFFECTS ....................................................................................................... 11

Interaction with the External Flow .......................................................................................... 12

Introducing Second-Order Effects .......................................................................................... 13

Substitution principle ............................................................................................................ 13

EXPERIMENTAL/NUMERICAL EFFORTS TO DATE ........................................................... 14

APPLICATION OF CFD ............................................................................................................ 17

DESCRIPTION OF CFD SOLVER (GASP) ............................................................................ 17

  Spatial Discretization of Equation (30) .............................................................................. 20

  Time Discretization of Equation (30) ................................................................................ 23

  Physical Modeling in GASP ............................................................................................... 25

  GASP Post-Processing ("PRINT") ....................................................................................... 26

ANALYSIS PARAMETERS ........................................................................................................ 27

  Boundary-layer Thickness, δ............................................................................................... 27

  Displacement Thickness, δ* ............................................................................................... 28

  Momentum thickness, θ ..................................................................................................... 28

  Shape Factor, H .................................................................................................................. 29

  Stanton Number (St) .......................................................................................................... 29

CFD SOLUTIONS TO STUDY EFFECTS ................................................................................ 29

FLOWFIELD CASES - DESCRIPTION ...................................................................................... 30
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>PHYSICAL GEOMETRY</td>
<td>30</td>
</tr>
<tr>
<td>BASELINE COMPUTATIONAL CASE (CASE1)</td>
<td>32</td>
</tr>
<tr>
<td>LINEAR STAGNATION ENTHALPY FUNCTIONS (CASE2)</td>
<td>33</td>
</tr>
<tr>
<td>NONLINEAR STAGNATION ENTHALPY FUNCTIONS (CASE3)</td>
<td>35</td>
</tr>
<tr>
<td>COMPUTATIONAL GRID DEVELOPMENT</td>
<td>37</td>
</tr>
<tr>
<td>BLASIUS BASELINE GRID</td>
<td>37</td>
</tr>
<tr>
<td>REFINED GRID INITIAL MODIFICATIONS</td>
<td>45</td>
</tr>
<tr>
<td>CASE2 RUNS WITH STRONGER GRADIENTS</td>
<td>49</td>
</tr>
<tr>
<td>GRID STANDARDIZATION EFFORTS</td>
<td>55</td>
</tr>
<tr>
<td>GRID ASPECT RATIO MODIFICATIONS / IDIMBC = -5 TESTS</td>
<td>63</td>
</tr>
<tr>
<td>&quot;idim&quot; Boundary Condition</td>
<td>63</td>
</tr>
<tr>
<td>Grid Extension Design</td>
<td>64</td>
</tr>
<tr>
<td>Fine/Coarse/Fine/Coarse Grid Design</td>
<td>64</td>
</tr>
<tr>
<td>Cell Aspect Ratio Analysis</td>
<td>66</td>
</tr>
<tr>
<td>Aspect Ratio Effects</td>
<td>66</td>
</tr>
<tr>
<td>Grid Compression Level Tests</td>
<td>68</td>
</tr>
<tr>
<td>Computational Runs (&quot;fncf&quot; Grid)</td>
<td>72</td>
</tr>
<tr>
<td>Uniform Grid Design</td>
<td>77</td>
</tr>
<tr>
<td>&quot;Visualization Layer&quot;</td>
<td>85</td>
</tr>
<tr>
<td>Uniform Grid With Vertical Sections (Velocity Oscillations)</td>
<td>86</td>
</tr>
<tr>
<td>Nonuniform Grid Design (&quot;Final&quot; Grid)</td>
<td>92</td>
</tr>
<tr>
<td>FLOWFIELD CASES - RESULTS</td>
<td>95</td>
</tr>
<tr>
<td>INVISID VALIDATION SOLUTION</td>
<td>95</td>
</tr>
<tr>
<td>Computation Description</td>
<td>95</td>
</tr>
<tr>
<td>Inviscid Results</td>
<td>95</td>
</tr>
<tr>
<td>CASE 1 (BASELINE UNIFORM)</td>
<td>96</td>
</tr>
<tr>
<td>Computation Description/Results</td>
<td>96</td>
</tr>
<tr>
<td>Comparison with Theory (Blasius Solution)</td>
<td>100</td>
</tr>
<tr>
<td>CASE 2 (LINEAR STAGNATION ENTHALPY GRADIENT, HO'=50.0)</td>
<td>101</td>
</tr>
<tr>
<td>Linear Gradient Description</td>
<td>101</td>
</tr>
<tr>
<td>Comparison with Baseline Uniform Flow (Case1)</td>
<td>103</td>
</tr>
<tr>
<td>Boundary-Layer Parameters</td>
<td>103</td>
</tr>
<tr>
<td>Skin Friction Coefficient/Stanton Number</td>
<td>105</td>
</tr>
<tr>
<td>CASE 3 (NONLINEAR STAGNATION ENTHALPY GRADIENT, σ=-0.029)</td>
<td>106</td>
</tr>
<tr>
<td>Nonlinear Gradient Description</td>
<td>106</td>
</tr>
<tr>
<td>Comparison with Baseline Uniform Flow (Case1)</td>
<td>107</td>
</tr>
<tr>
<td>Boundary-Layer Parameters</td>
<td>107</td>
</tr>
<tr>
<td>Skin-Friction Coefficient/Stanton Number</td>
<td>109</td>
</tr>
<tr>
<td>REDUCED REYNOLDS NUMBER COMPUTATIONS</td>
<td>110</td>
</tr>
<tr>
<td>Computation Description</td>
<td>110</td>
</tr>
<tr>
<td>Case 1 Results</td>
<td>111</td>
</tr>
<tr>
<td>Pressure Profiles</td>
<td>112</td>
</tr>
<tr>
<td>Density Profiles</td>
<td>112</td>
</tr>
<tr>
<td>Temperature Profiles</td>
<td>113</td>
</tr>
<tr>
<td>Case2 Comparison with Baseline Uniform Flow (Case1)</td>
<td>114</td>
</tr>
<tr>
<td>Pressure Profiles</td>
<td>114</td>
</tr>
<tr>
<td>Density Profiles</td>
<td>115</td>
</tr>
<tr>
<td>Temperature Profiles</td>
<td>116</td>
</tr>
<tr>
<td>CONCLUSIONS</td>
<td>118</td>
</tr>
<tr>
<td>RECOMMENDATIONS</td>
<td>121</td>
</tr>
</tbody>
</table>
LIST OF FIGURES

Figure 1. Treatment of Boundary Cells in GASP .......................................................... 18
Figure 2. Physical Model Mapping to Computational Space ................................. 26
Figure 3. Initial Baseline Physical Geometry ............................................................ 31
Figure 4. Case1 Physical Geometry ............................................................................ 32
Figure 5. Case2 Physical Geometry ............................................................................ 33
Figure 6. Case3 Physical Geometry ............................................................................ 35
Figure 7. Blasius Grid ................................................................................................. 38
Figure 8. Case1 $u$ Velocity Contour Plot - (41x21) Size Grid ......................... 40
Figure 9. Case1 $u$ Velocity Contour Plot - (61x21) Size Grid ......................... 41
Figure 10. Comparison of Case1 $\delta$ parameter - (41x21) and (61x21) Size Grids ................................................................. 42
Figure 11. Enthalpy and Velocity Gradient with $h_0^{'^0}=0.5$ - (61x21) Size Grid $y_{\text{max}}=0.001$ (m) ......................... 43
Figure 12. Boundary-Layer Parameters Case1/Case2 $h_0^{'}=0.5$ - (61x21) Size Grid ........................................................................... 44
Figure 13. Refined Grid After Initial Modifications (61x101) Size Grid $y_{\text{max}}=0.005$ (m) ................................................................. 46
Figure 14. Enthalpy and Velocity Gradient with $h_0^{'}=10.0$ (61x101) Size Grid $y_{\text{max}}=0.005$ (m) ................................. 47
Figure 15. Boundary-Layer Parameters Case1/Case2 $h_0^{'}=10.0$ - (61x101) Size Grid ........................................................................... 48
Figure 16. Enthalpy and Velocity Gradient with $h_0^{'}=100.0$ (61x101) Size Grid $y_{\text{max}}=0.005$ (m) ................................. 49
Figure 17. Case2 $u$ Velocity Contour Plot on (61x101) "Standardized Grid" ................................................................. 50
Figure 18. Boundary-Layer Parameters Case2 Space/Case2 Time $h_0^{'}=100.0$ - (61x101) Size Grid ........................................................................... 52
Figure 19. Boundary-Layer Parameters Case1/Case2 Space-Time $h_0^{'}=100.0$ - (61x101) Size Grid ................................................................. 54
Figure 20. Case1 $u$ Velocity Contour Plot on (61x101) "Standardized Grid" ................................................................. 56
Figure 21. Linear Stagnation Enthalpy Gradient and Velocity Profile at $h_0^{'}=1000.0$ ........................................................................... 57
Figure 22. Case2 $u$ Velocity Contour Plot on (61x101) "Standardized Grid", $h_0^{'}=1000.0$ ........................................................................... 58
Figure 23. Linear Stagnation Enthalpy Gradient and Velocity Profile at $h_0^{'}=3000.0$ ........................................................................... 59
Figure 24. Case2 $u$ Velocity Contour Plot on (61x101) "Standardized Grid", $h_0^{'}=3000.0$ ........................................................................... 60
Figure 25. Case3 $u$ Velocity Contour Plot on (61x101) "Standardized Grid", $\sigma=-100$ ........................................................................... 61
Figure 26. Velocity Profiles Showing Flow Oscillations at Six "k" Stations ........................................................................... 62
Figure 27. F/C/F/C Grid Design Diagram with Break-Points ........................................................................... 65
Figure 28. Definition of Poor Aspect Ratio Cells ........................................................................... 66
Figure 29. Example of Poor Aspect Ratios in Current Grid Design ........................................................................... 67
Figure 30. Grid Compression Test1 Mesh ($\beta=1.0005$) ........................................................................... 70
Figure 31. $u$ Velocity Contour Near Stagnation Point for Compression Tests ........................................................................... 70
Figure 32. Case2 $u$ Velocity Contour Plot on (101x401) "Standardized Grid", $h_0^{'}=1000.0$ ........................................................................... 71
Figure 33. General Shape of Gradients with Constant $h_0$ for Top 20% $y_{\text{max}}$ ........................................................................... 72
Figure 34. Linear Stagnation Enthalpy Gradient and Velocity Profile at $h_0^{'}=3000.0$ ........................................................................... 73
Figure 35. $u$ Velocity Contour Plot for Case2, $h_0^{'}=1000.0$ ........................................................................... 73
Figure 36. Stagnation Enthalpy and Velocity Gradient with $h_0^{'}=25$ ........................................................................... 74
Figure 37. $u$ Velocity Contour Plot for Case2, $h_0^{'}=25$ (idimbc=-3) ........................................................................... 75
Figure 38. Enthalpy and Velocity Gradient with $h_0^{'}=25$, $y_{\text{max}}=0.1$ m ........................................................................... 76
Figure 39. Enthalpy and Velocity Gradient with $h_0^{'}=1000.0$ on Uniform Grid ........................................................................... 79
Figure 40. Velocity Vector Plot, Case2 $h_0^{'}=1000.0$ ........................................................................... 80
Figure 41. Velocity Vector Plot with $i_0$ Boundary Blow-up, Case1 ........................................................................... 80
Figure 42. Enthalpy and Velocity Gradient with $h_0^{'}=50.0$ on Uniform Grid ........................................................................... 81
Figure 43. $u$ Velocity Contour Plot (Case2, $h_0^{'}=50.0$) ........................................................................... 82
Figure 44. Enthalpy and Velocity Gradient with \( h'_o = 5.0 \) on Uniform Grid ................................................................. 83
Figure 45. \( u \) Velocity Contour Plot (Case2, \( h'_o = 5.0 \)) ................................................................. 84
Figure 46. Uniform Computational Grid Points Within Boundary-Layer ......................................................... 85
Figure 47. Comparison of TECPLLOT Contours w/Node & Cell Interpolation ........................................... 86
Figure 48. Enthalpy and Velocity Gradient with \( h'_o = 5.0 - \text{BLHGT}=0.003 \text{ (m)} / \text{BLPTS}=11 \) ........ 88
Figure 49. Enthalpy and Velocity Gradient with \( h'_o = 0.5 - \text{BLHGT}=0.0015 \text{ (m)} / \text{BLPTS}=11 \) ........ 89
Figure 50. Enthalpy and Velocity Gradient with \( h'_o = 0.2 - \text{BLHGT}=0.0015 \text{ (m)} / \text{BLPTS}=11 \) ........ 90
Figure 51. \( u \) Velocity Contour Plot for Case2 with \( h'_o = 0.2 - \text{BLHGT}=0.0015 \text{ (m)} / \text{BLPTS}=11 \) ..... 91
Figure 52. \( u \) Velocity Contour Plot for Case1 ......................................................................................... 91
Figure 53. Final Grid Design for All Cases ................................................................................................. 93
Figure 54. \( u \) Velocity Contour Plot for Inviscid Validation Solution ...................................................... 96
Figure 55. \( u \) Velocity Contour Plot for Case1 on Small Domain "Final" Grid ......................................... 97
Figure 56. \( u \) Velocity Contour Plot for Case1 on Large Domain "Final" Grid .......................................... 97
Figure 57. Modified Final Grid Design for All Cases (\( \text{ymax}=0.002 \text{ (m)} \)) ........................................ 98
Figure 58. Skin Friction Coefficient Along Plate Surface (Case1) ............................................................ 99
Figure 59. Heat Transfer Coefficient (St) Along Plate Surface (Case1) ..................................................... 100
Figure 60. Comparison of Case1 Results with Blasius Velocity Profile ................................................... 101
Figure 61. Enthalpy and Velocity Gradient with \( h'_o = 50.0 \) on Modified "Final" Grid ............................ 102
Figure 62. \( u \) Velocity Contour Plot for Case2 \( h'_o = 50 \) on Modified "Final" Grid ............................ 103
Figure 63. Comparison of Boundary-Layer Parameters in Analysis Region (Case1/Cas2) ............... 104
Figure 64. Skin Friction Coefficient Along Plate Surface (Case1 vs. Case2) ...................................... 105
Figure 65. Stanton Number Along Plate Surface (Case1 vs. Case2) ...................................................... 106
Figure 66. Enthalpy and Velocity Gradient with \( \sigma=-0.026 \) on Modified "Final" Grid .......................... 107
Figure 67. Comparison of Case3/Case1 Boundary-Layer Parameters .................................................... 108
Figure 68. Skin Friction Coefficient Along Plate Surface (Case1 vs. Case3) ........................................... 109
Figure 69. Stanton Number Along Plate Surface (Case1 vs. Case3) ...................................................... 110
Figure 70. Reduced Re# Computational Grid ......................................................................................... 111
Figure 71. Reduced Re# Case1 Pressure Profiles ................................................................................... 112
Figure 72. Reduced Re# Case1 Density Profiles ..................................................................................... 113
Figure 73. Reduced Re# Case1 Temperature Profiles ............................................................................. 114
Figure 74. Comparison of Reduced Re# Case1 and Case2 Pressure Profiles ...................................... 115
Figure 75. Comparison of Reduced Re# Case1 and Case2 Density Profiles .................................. 116
Figure 76. Comparison of Reduced Re# Case1 and Case2 Temperature Profiles ................................ 117
LIST OF SYMBOLS

(Listed in order of use)

\( u \)  
tangential velocity component

\( v \)  
normal velocity component

\( x \)  
coordinate streamwise along the plate

\( y \)  
coordinate normal to the plate

\( z \)  
coordinate spanwise to the plate

\( P \)  
local pressure

\( \rho \)  
local density

\( \mu \)  
local viscosity

\( \tilde{y} \)  
coordinate normal to the plate within the boundary-layer

\( \tilde{v} \)  
normal velocity component within the boundary-layer

\( \varepsilon \)  
perturbation parameter

\( \mu_{\infty} \)  
freestream viscosity

\( \rho_{\infty} \)  
freestream density

\( U_{\infty} \)  
freestream velocity

\( l \)  
length of plate

\( Re \)  
Reynolds Number

\( \mathcal{S} \)  
mathematical order

\( t \)  
local temperature

\( R \)  
local density in outer region

\( \gamma \)  
ratio of specific heats

\( M_{\infty} \)  
freestream Mach number

\( h \)  
local enthalpy

\( w \)  
velocity vector

\( \lambda \)  
arbitrary transformation function

\( i \)  
computational coordinate in \( x \) direction

\( j \)  
computational coordinate in \( z \) direction

\( k \)  
computational coordinate in \( y \) direction

\( i_{\text{dim}} \)  
maximum grid dimension in \( i \) direction

\( j_{\text{dim}} \)  
maximum grid dimension in \( j \) direction

\( k_{\text{dim}} \)  
maximum grid dimension in \( k \) direction

\( z \)  
coordinate spanwise along the plate

\( Q \)  
vector of conserved flow variables

\( \tilde{F} \)  
inviscid flux vector

\( \tilde{F}_v \)  
viscous flux vector

\( \hat{n} \)  
unit vector normal to cell face
\( d\mathcal{A} \) area of cell face
\( d\mathcal{V} \) volume of cell
\( \Phi \) vector of source terms
\( q \) vector of primitive flow variables
\( e \) species internal energy per unit mass
\( K, \varphi_K \) turbulent kinetic energy
\( \varepsilon, \varphi_\varepsilon \) dissipation rate of turbulent kinetic energy
\( \kappa \) parameter for spatial accuracy of inviscid and viscous flux interpolation
\( \varphi \) flag for higher-order accurate inviscid and viscous flux interpolation
\( R(q) \) residual grouping of all flux terms
\( k \) GASP spatial interpolation accuracy control parameter
\( \mu \) laminar viscosity
\( E_k, F_i \) Sutherland empirical coefficients
\( \hat{\mathcal{F}}, \hat{\mathcal{G}}, \hat{\mathcal{H}} \) inviscid flux vectors in \( i,j,k \) directions, respectively
\( \hat{\mathcal{F}}_v, \hat{\mathcal{G}}_v, \hat{\mathcal{H}}_v \) viscous flux vectors in \( i,j,k \) directions, respectively
\( i_0 \) minimum grid dimension in \( i \) direction
\( j_0 \) minimum grid dimension in \( j \) direction
\( k_0 \) minimum grid dimension in \( k \) direction
\( c_f \) skin friction coefficient
\( S_t \) Stanton number
\( \delta \) boundary-layer thickness
\( \delta^* \) displacement thickness
\( \theta \) momentum thickness
\( H \) shape factor
\( h \) heat transfer
\( c_p \) specific heat constant pressure
\( \text{Pr} \) Prandtl number
\( \text{Re}_k \) local Reynolds number
\( \text{kObc} \) \( k_0 \) boundary condition
\( \text{Kdimbc} \) \( k_d \) boundary condition
\( \text{iObc} \) \( i_0 \) boundary condition
\( \text{idimbc} \) \( i_d \) boundary condition
\( h_o \) stagnation enthalpy
\( h_o' \) stagnation enthalpy gradient
\( \sigma \) error function shaping parameter
\( \text{erf} \) error function
\( \beta \) computational grid exponential stretching parameter
\( \chi_{\text{max}} \) maximum \( x \) grid value
\( \chi_{\text{max}} \) maximum \( y \) grid value
\( \text{AR} \) computational grid aspect ratio
\( \Delta y \) cell height
\( \Delta x \) cell width/space step
\( \Delta t \) time step
\( c \) wave speed
\( \lambda_\alpha, \lambda_\gamma \) acoustic eigenvalues in \( x \) and \( y \) direction, respectively
INTRODUCTION

In 1904, Ludwig Prandtl developed first-order boundary-layer theory as a simplification of the Navier-Stokes equations. He realized that since the boundary-layer is very small at large Reynolds numbers, the assumptions of

\[ \nu \ll u \quad \text{and} \quad \frac{\partial}{\partial x} \ll \frac{\partial}{\partial y} \]  

Equ. (1)

are valid, where \( u \) and \( \nu \) are the tangential and normal velocity components, respectively and \( x \) is the flow direction, \( y \) the normal direction. Therefore, terms of such magnitude can be neglected from the full Navier-Stokes equations. This simplification leads to the first-order boundary-layer equations. While first-order boundary-layer theory provides reasonable solutions for most engineering problems, the omission of the higher-order terms eliminates certain important second-order effects. For example, if the flow outside of a boundary-layer has an entropy gradient, first-order boundary-layer theory ignores its effects on the flow. However, the effects can be captured with second-order boundary-layer theory. A typical example where entropy gradients effect the boundary-layer flow and heat transfer is found in a scramjet. For compressible flow, Table 1 gives the phenomena that are not accounted for with first-order theory. \(^3\)
Table 1. Second-Order Effects for Compressible Flow

A. Curvature
   1. Longitudinal
   2. Transverse
B. Interaction with the external flow
   3. Displacement
   4. External entropy gradient
   5. External stagnation enthalpy gradient
C. Noncontinuum surface effects
   6. Velocity slip
   7. Temperature jump

Few validating comparisons have been attempted between second-order boundary-layer theory and experimental results or numerical Navier-Stokes solutions of compressible viscous flows. Experimental simulations to capture second-order effects are difficult since the desired effects are small and can exist simultaneously. Also, there is the possibility that simultaneously existing effects may have a neutralizing influence on each other, reducing the overall flow response. 4 In 1969, Van Dyke proposed the following evaluation of the status of boundary-layer theory at that time;

Convincing quantitative experimental confirmation (or refutation) of the validity of higher-order boundary-layer theory has not yet been achieved. Measurements are meager at low speeds, where the theory is nearly unassailable. At high speeds, where we have seen that kinetic theory casts some doubt upon even the second approximation, experimental data are more numerous but often in disagreement. Nevertheless, most experiments seem to show at least qualitative accord with the predictions of second-order theory. 5

Emanuel goes further to suggest that Van Dyke’s statement is true today, representing a serious lack of confirmation of the applicability and limitations of second-order boundary-layer theory. 6

Application of Computational Fluid Dynamics (CFD) to the solution of boundary-layer flows provides a flexible experimental resource. Using CFD, it is feasible to solve the full Navier-Stokes equations to capture the second-order boundary-layer effects of interest. This report is a discussion of the numerical experimentation performed to capture and analyze the second-order boundary-layer effects of a supersonic flow over a flat plate at zero angle of attack. The second-order effects are initiated by the introduction of a stagnation enthalpy gradient in the flowfield at the plate leading
edge. The limitations and applicability of the second-order theory are tested through the use of several different nonuniform flowfields, both linear and nonlinear.
SECOND-ORDER BOUNDARY-LAYER THEORY

Perturbation Methods (Matched Asymptotic Expansions)

The theory of Perturbation Methods allows the development of an approximate solution for any set of algebraic, differential or integral equations. The approach is to expand the relevant variables in a power series of a new parameter. The expansions are then substituted into the original equations, and the resulting approximation is evaluated for uniformity. It is this approach that leads to the second-order boundary-layer equations.

The overall intent of second-order boundary-layer theory involves developing an approximate solution to the Navier-Stokes equations which is accurate to second-order for boundary-layer flows. We seek a unique set of higher-order governing equations for the inner flow region (the boundary-layer), and a separate set of equations for the outer flow region (the external flow). The interface of these regions is handled through the selection of matched boundary conditions, which satisfy both sets of equations.

For each region, two sets of equations are developed, including a first-order and a second-order solution. The first-order solution is necessary for evaluation of the second-order equations. The development of the theory as presented here is consistent with the Perturbation Methods of Van Dyke as applied by Emanuel. The discussion is intended as an overview and the reader is urged to consult the references for a detailed derivation and explanation.

To begin, we consider the conservation form of the two-dimensional Navier-Stokes equations, and assume perfect gas conditions, which gives;
\[ \frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) = 0 \]  
Equ. (2a)

\[ \rho \left( u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = -\frac{\partial p}{\partial x} + \mu \left( 2 \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 v}{\partial x \partial y} \right) \]  
Equ. (2b)

\[ \rho \left( u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} \right) = -\frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 u}{\partial x y} + 2 \frac{\partial^2 v}{\partial y^2} \right) \]  
Equ. (2c)

where \( \rho \) is the density and \( P \) is the pressure. Within the boundary-layer, the variables \( y \) (distance normal to plate) and \( v \) (normal velocity component) will be very small compared to the same variables in the external flow region. Therefore we can scale \( y \) and \( v \), distinguishing the inner \( y \) and \( v \) variables with a bar notation;

\[ y = \varepsilon \bar{y} \quad v = \varepsilon \bar{v} \]  
Equ. (3)

where \( \varepsilon \) is the small perturbation parameter defined as;

\[ \varepsilon = \left( \frac{\mu_{\infty}}{\rho_{\infty} U_{\infty}} \right)^{1/2} = \frac{1}{Re^{1/2}} \]  
Equ. (4)

In Equation (4) the \( \infty \) symbol represents freestream flow conditions, \( \mu \) is the viscosity, \( l \) is the plate length, \( U \) is the velocity and \( Re \) is the Reynolds number. The scaled variables are substituted into Equations (2), and only terms of \( \mathcal{O}(1) \) and \( \mathcal{O}(\varepsilon) \) are kept, where the \( \mathcal{O} \) symbol denotes order. The result of this substitution is the combined first and second-order boundary-layer equations. When the perturbation parameter, \( \varepsilon \), is taken as zero the equations reduce to the first-order boundary-layer equations.\(^9\)

**Inner Region (Boundary-Layer Equations)**

Obtaining a second-order accurate form of the boundary-layer equations requires only including terms of integer powers of \( \varepsilon \). With that requirement we asymptotically expand the inner flow variables, representing first-order flow variables with a 1 subscript and second-order variables with a 2 subscript.\(^{10}\) Small letters distinguish inner variables while capitals represent outer variables.
The flow parameters are consistent with previous definitions, with the addition of $t$ and $T$ for inner and outer temperature respectively. The inner expansion gives:

\begin{align*}
    u(x,y;\varepsilon) &\sim u_1(x,y) + \varepsilon u_2(x,y) + \cdots & \text{Equ. (5a)} \\
    v(x,y;\varepsilon) &\sim v_1(x,y) + \varepsilon v_2(x,y) + \cdots & \text{Equ. (5b)} \\
    p(x,y;\varepsilon) &\sim p_1(x,y) + \varepsilon p_2(x,y) + \cdots & \text{Equ. (5c)} \\
    \rho(x,y;\varepsilon) &\sim \rho_1(x,y) + \varepsilon \rho_2(x,y) + \cdots & \text{Equ. (5d)} \\
    T(x,y;\varepsilon) &\sim T_1(x,y) + \varepsilon T_2(x,y) + \cdots & \text{Equ. (5e)}
\end{align*}

\[
\mu(x,y;\varepsilon) = \mu(T) = \mu(t_1 + \varepsilon t_2 + \cdots) - \mu(t_1) \frac{d\mu}{dt} \varepsilon t_2 + \cdots = \mu_1 + \varepsilon \mu_2 + \cdots
\]

Equ. (5f)

These variable expansions, Equations (6), are substituted into the combined boundary-layer equations.

For the first-order equations we retain only the first-order terms, and arrive at the standard, first-order boundary-layer equations for a perfect gas. The second-order inner equations require that we keep the first and second-order variable terms, with all terms of order higher than $\Theta(\varepsilon)$ dropped.

**Outer Region (External Flow Equations)**

The external flow region is treated in much the same way as the inner flow region. We expand the outer flow variables asymptotically, retaining only integer powers for $\varepsilon$:

\begin{align*}
    u(x,y;\varepsilon) &\sim U_1(x,y) + \varepsilon U_2(x,y) + \cdots & \text{Equ. (6a)} \\
    v(x,y;\varepsilon) &\sim V_1(x,y) + \varepsilon V_2(x,y) + \cdots & \text{Equ. (6b)} \\
    p(x,y;\varepsilon) &\sim P_1(x,y) + \varepsilon P_2(x,y) + \cdots & \text{Equ. (6c)} \\
    \rho(x,y;\varepsilon) &\sim \rho_1(x,y) + \varepsilon \rho_2(x,y) + \cdots & \text{Equ. (6d)} \\
    T(x,y;\varepsilon) &\sim T_1(x,y) + \varepsilon T_2(x,y) + \cdots & \text{Equ. (6e)}
\end{align*}

where $\rho$ represents the density of the outer region. The variable expansions are substituted in the combined boundary-layer equations and the first and second-order equations are obtained for the external region.
Boundary Conditions

**Inner Equations (First-Order)**

Given the physical assumptions associated with boundary-layer flow, we can develop the boundary conditions at the wall for the inner flow region. For the inner flow, the assumption of a no-slip, constant temperature, solid wall provides the first-order boundary conditions at the plate surface ($y=0$);

\[ \nu_1(x,0) = 0 \quad \text{Equ. (7a)} \]
\[ u_1(x,0) = 0 \quad \text{Equ. (7b)} \]
\[ t_1(x,0) = T_{\text{wall}}(x) \quad \text{Equ. (7c)} \]

**Inner Equations (Second-Order)**

The boundary conditions for the second-order inner equations are developed by substituting the inner variable expansions, Equations (5), into the known velocity slip and temperature jump conditions at the wall;

\[ u(x,0) = \varepsilon \gamma \frac{1}{2} M_\infty \left[ \frac{\mu}{\rho} \left( a_1 \gamma \frac{1}{2} \frac{\partial u}{\partial x} + \frac{3}{4 \gamma \frac{1}{2} M_\infty} \frac{\partial T}{\partial y} \right) \right]_{\text{wall}} \quad \text{Equ. (8a)} \]
\[ T(x,0) = T_{\text{wall}} + c_1 \varepsilon \gamma \frac{1}{2} M_\infty \quad \text{Equ. (8b)} \]

where $a_1$ is a velocity dependent coefficient, $c_1$ is a temperature based coefficient, $\gamma$ is the ratio of specific heats, and $M$ is the Mach number. Substituting the expansions and scaling the inner variables with Equation (3) gives the following boundary conditions for the second-order inner terms;
The substitution also gives the first-order inner boundary conditions at the wall, which we determined earlier using physical flow assumptions in Equations (7). The no-slip solid flat plate assumption also gives:

\[ \nu_2(x,0) = 0 \]  
\text{Equ. (10)}

**Outer Equations (First-Order)**

The first-order outer boundary conditions are developed from assumptions based on the physical model, as was done for the inner equations. The solid flat plate, no-slip assumption provides:

\[ V_1(x,0) = 0 \]  
\text{Equ. (11)}

This condition is proven mathematically in the development of the interface matching conditions, presented later. The other boundary condition is found as \( y \) approaches the freestream flow, where the first-order variables must satisfy the freestream conditions, giving:

\[ u_1(x,\infty) \rightarrow U_\infty, \quad \nu_1(x,\infty) \rightarrow V_\infty, \quad t_1(x,\infty) \rightarrow T_\infty \ldots \]  
\text{Equ. (12)}

**Outer Equations (Second-Order)**

Finally, the boundary conditions for the second-order outer equations are developed for the freestream and plate surface boundaries. From Equation (12), the first-order outer boundary conditions approach freestream values in the outer flow region. Therefore, the second-order flow variables will be reduced to zero given the absence of second-order effects in the freestream flow, giving:

\[ U_2(x,\infty) \rightarrow 0, \quad P_2(x,\infty) \rightarrow 0, \quad T_2(x,\infty) \rightarrow 0 \ldots \]  
\text{Equ. (13)}

At the plate surface, the following condition is found for the second-order normal velocity component, \( V_2 \):
This condition is not obvious from the physical flow model. However, a mathematical derivation is presented in the following section, "Interface Matching Conditions (Inner Variables)".

Interface Matching Conditions (Inner Variables)

The Perturbation Method ultimately provides a unique second-order flow solution for both the inner and outer regions. The first-order solution is obtained and used to solve the second-order equations in each region. However, the solution at the interface between the regions, the boundary-layer outer edge, must be addressed separately. Applying the Matching Principle provides the boundary conditions needed to evaluate the flow solution at the interface. The approach is warranted because the validity of the asymptotic expansions overlaps the interface for both the inner and outer regions.\(^\text{12}\)

The application of the Matching Principle requires the development of meaningful boundary conditions for both solution regions. For all of the flow variables, we match the inner and outer expansions at the interface. The matching can be performed in terms of either the inner or outer variables, however for simplicity we follow the convention of Emanuel, matching in terms of the inner variables.\(^\text{13}\)

From Equation (5a) for the inner variable expansions we have \(u\) as;

\[
u(x,y;\epsilon) \sim u_1(x,y) + \epsilon u_2(x,y) + \cdots \quad \bar{y} \to \infty \quad \text{Equ. (15)}
\]

where \(\infty\) refers to the boundary-layer outer edge. The outer expansion is expressed in a Taylor Series about \(y=0\) with each term expanded independently, and then transformed to inner region variables, using Equation (3), which gives;\(^\text{14}\)

\[
u \sim U_1(x,0) + \epsilon \bar{y} \frac{\partial U_1}{\partial y}(x,0) + \epsilon^2 \frac{\bar{y}^2}{2} \frac{\partial^2 U_1}{\partial y^2}(x,0) + \cdots
\]

\[
+ \epsilon U_2(x,0) + \epsilon^2 \bar{y} \frac{\partial U_2}{\partial y}(x,0) + \cdots
\quad \text{Equ. (16)}
\]
Comparison of "like-ordered" terms (9(1) and 9(é)) in Equations (15) and (16) gives;

\[ u_1(x, \infty) = U_1(x,0) \quad \text{Equ. (17a)} \]

\[ u_2(x, \bar{y}) \sim \bar{y} \frac{\partial U_1}{\partial y}(x,0) + U_2(x,0) \quad y \to \infty \quad \text{Equ. (17b)} \]

The same method is used for the other flow variables, yielding;

**9(1) Terms**

\[ p_1(x, \infty) = P_1(x,0) \]

\[ \rho_1(x, \infty) = \rho_1(x,0) \quad \text{Equ. (18a)} \]

\[ t_1(x, \infty) = T_1(x,0) \]

and,

**9(é) Terms**

\[ p_2(x, \bar{y}) \sim \bar{y} \frac{\partial P_1}{\partial y}(x,0) + P_2(x,0) \quad y \to \infty \]

\[ \rho_2(x, \bar{y}) \sim \bar{y} \frac{\partial \rho_1}{\partial y}(x,0) + \rho_2(x,0) \quad y \to \infty \quad \text{Equ.(18b)} \]

\[ t_2(x, \bar{y}) \sim \bar{y} \frac{\partial T_1}{\partial y}(x,0) + T_2(x,0) \quad y \to \infty \]

The remaining matching condition at the interface that we seek is for the normal velocity component, \( v \). However, for the inner region, the expansion for \( v \), Equation (5b), was performed in terms of inner variable \( \tilde{v} \). To properly match the boundary conditions at the interface we need a condition for \( v \).

So, substituting the scaling for \( v \) from Equation (3) in Equation (5b) we rewrite the inner \( v \) expansion as;

\[ v(x, y; \epsilon) \sim \epsilon v_1(x, \bar{y}) + \epsilon^2 v_2(x, \bar{y}) + \cdots \quad \bar{y} \to \infty \quad \text{Equ. (19)} \]

The outer expansion is again expressed in a Taylor series and the height, \( y \), scaled to inner variable \( \bar{y} \), giving;
\[ v \sim V_1(x,0) + \varepsilon \overline{y} \frac{\partial V_1}{\partial y}(x,0) + \varepsilon^2 \frac{\overline{y}^2}{2} \frac{\partial^2 V_1}{\partial y^2}(x,0) + \ldots \]  
Equ. (20)

\[ + \varepsilon V_2(x,0) + \varepsilon^2 \overline{y} \frac{\partial V_2}{\partial y}(x,0) + \ldots \]

Comparison of “like-ordered” terms (\(S(1)\) and \(S(\varepsilon)\)) gives;

\[ V_1(x,0) = 0 \]  
Equ. (21)

\[ v_1(x,\overline{y}) \sim \overline{y} \frac{\partial V_1}{\partial y}(x,0) + V_2(x,0) \quad y \to \infty \]  
Equ. (22)

The problem does not require the first-order boundary condition for \(v_1\) at the interface, Equation (22).

However, Equation (22) can be used to develop the second-order boundary condition at the wall for the outer flow region, \(V_2(x,0)\). Differentiating the first-order term \((v_1\) or \(V_1\)) with respect to \(\overline{y}\) in the original inner and outer expansions, Equations (5) and (6), we find;

\[ \frac{\partial V_1}{\partial y}(x,0) = \frac{\partial v_1}{\partial \overline{y}}(x,\infty) \]  
Equ. (23)

Substituting Equation (23) into Equation (22) provides the following form of \(V_2(x,0)\);

\[ V_2(x,0) \sim v_1(x,\overline{y}) - \overline{y} \frac{\partial v_1}{\partial \overline{y}} \quad \overline{y} \to \infty \]  
Equ. (24)

Equation (24) serves as the second-order boundary condition for the normal velocity component at the plate surface, as presented previously in Equation (14).

Thus we have found the matching conditions for the inner equations at the boundary-layer outer edge. The \(S(1)\) terms, Equations (17a) and (18a) are the first-order conditions. The \(S(\varepsilon)\) terms in Equations (17b), and (18b) can be rewritten by evaluating the flow gradients at the wall \((x,0)\), providing the second-order inner matching conditions at \(y \to \infty\), the flow interface.

**Second-Order Effects**

As discussed in the introduction, certain real-world engineering flow problems are critically sensitive to second-order effects. Sawley and Wuthrich observed that for hypersonic flows over re-entry vehicles, the flow behind the ensuing bow shock wave is in chemical non-equilibrium. A
complete solution of the three-dimensional Navier-Stokes equations is not feasible and the application
of high-order, computationally inexpensive methods is therefore of interest. Goldstein, Lieb and
Cowley investigated the boundary-layer effects of small imperfections in the free-stream flow. Such
imperfections can occur in experimental studies, particularly boundary-layer transition experiments.

In such cases, application of the first-order boundary-layer theory is inadequate. However, for simplified, quick analysis situations full blown numerical solutions of the Navier-Stokes equations may not be justified. Therefore, second-order boundary-layer theory is a useful analytical tool which is merited in certain engineering situations.

Interaction with the External Flow

The second-order effect of primary interest to this study is the interaction of the boundary-
layer with the external flow. The displacement of the external flow by the boundary-layer through a
distance \( \delta^* \) (the displacement thickness) is a common characteristic studied in first-order analysis.

Solution of the first-order boundary-layer equations neglects the effects of this displacement on the flow. We would expect the second-order equations to capture the influence of the displacement effect on the boundary-layer and external flow. However, the second-order effects are not independent and thus flow anomalies may be attributable to more than one of the effects in Table 1. As an example, the stagnation enthalpy gradient and the displacement will both affect the plate skin friction and heat transfer. For the current analysis we seek only to numerically solve the given flow and quantify the resulting higher-order effects. Isolating the second-order effect which caused a given flow response is not critical.

Comparing the first-order, two-dimensional boundary-layer equations, Equations (25a-b) with the Navier-Stokes equations, Equations (2a-c), we can see where the boundary-layer equations fail to capture second-order flow effects;

\[
\frac{\partial}{\partial x} (\rho u) + \frac{\partial}{\partial y} (\rho v) = 0 \quad \text{Equ. (25a)}
\]
The influence of rotational flow is neglected since there is no mathematical representation to account for the momentum of fluid particles in the normal direction. Additionally, the x-direction momentum equation is naturally missing all terms of order higher than $O(1)$ and thus second-order flow effects accounted for through those terms are neglected.

A curved shock wave upstream can result in rotational, nonhomentropic flow downstream, producing an external entropy gradient. Additionally, nonuniform heat addition or combustion produces nonhomenergetic inviscid flow, resulting in a stagnation enthalpy gradient. With first-order boundary-layer theory, neither of these gradient effects is captured in the flow solution. However, the CFD analysis should reveal any flow responses due to the effects since those computations will solve the Navier-Stokes equations.

Introducing Second-Order Effects

For the current analysis we focus on the interaction of the boundary-layer with the external flow. To induce flow interaction effects we setup a test case with the proper initial flow conditions. Nonuniform heat addition upstream in the flowfield will result in a stagnation enthalpy gradient downstream leading to flow interaction effects. Therefore, introducing a stagnation enthalpy gradient in the flowfield at the leading edge of the plate will provide the proper conditions for a strong second-order boundary-layer flow.

Emanuel provides an approach to solving the second-order boundary-layer equations for the flat plate geometry studied in the current analysis. His method uses the substitution principle to develop the first-order outer flow. We also apply the substitution principle to develop the nonuniform freestream flow.

Substitution principle

The Substitution principle is the method used to characterize the stagnation enthalpy gradient and introduce it in the initial flowfield. The approach is to transform the baseline, simple uniform

\[
\rho \left( u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = -\frac{d}{dx} + \mu \frac{\partial^2 u}{\partial y^2}
\]

Equ. (25b)
flowfield into a new, nonuniform, rotational, nonhomenergetic flow. Mathematically, the relationship between the baseline flowfield and the transformed flow is denoted by;

\[ p = \lambda^\phi p_b \quad \rho = \lambda^\alpha \rho_b \quad h = \lambda^\psi h_b \quad w_i = \lambda^\zeta w_{bi} \quad \text{Eq. (26)} \]

where \( h \) is represents the flow enthalpy, the \( b \) subscript signifies baseline flow variables, \( w_i \) is the velocity vector, the \( c_i \) denotes presently unknown constants and the \( \lambda \) parameter is an arbitrary transformation function. Assuming \( \lambda \) is constant along the flow streamlines, the \( c_i \) constants are determined by substituting the transformed flow variables into the steady Euler equations. The result is

\[ p = p_b \quad \rho = \lambda^{-1} \rho_b \quad h = \lambda \ h_b \quad w_i = \lambda^\zeta w_{bi} \quad \text{Eq. (27)} \]

Now for the problem at hand the flowfield is parallel, providing the convenient simplification,

\[ w_2 = w_3 = 0 \quad \text{Eq. (28)} \]

Therefore, the flowfield velocity transformation for a parallel flow, is given simply by;

\[ u = \lambda^{1/2} u_b = \lambda^{1/2} U_w \quad \text{Eq. (29)} \]

where the \( w_1 \) velocity component is replaced with the more familiar form of the \( x \) velocity component, \( u \).

**Experimental/Numerical Efforts to Date**

The origin of second-order boundary-layer theory dates to Prandtl’s 1935 observation that improvements could be made to the first-order Blasius solution through approximations for successive levels of accuracy:

Instead of the simple parallel flow, the flow around a parabolic cylinder of thickness 2\( \delta \) should be introduced, which would slightly alter the pressure distribution. The...calculation would have to be repeated for this new pressure distribution and if necessary the process repeated on the basis of the new measure of displacement so obtained.\(^{19}\)

The application of Perturbation Methods to the development of an approximation for the Navier-Stokes equations provided the key to the solution of the second-order boundary-layer
equations. Early efforts with the method of matched asymptotic expansions resulted in the standard approach, as presented here.

Numerical solutions initially focused on incompressible flows. Early efforts in compressible flow solutions include those of Kuo who numerically solved the supersonic, compressible boundary-layer flow for a flat plate. He showed the effects of the leading-edge shock wave on the downstream flow, exhibiting good correlation of the pressure distribution with the limited experimental data of the time. Thomann studied the turbulent boundary-layer flow with concave curvature, identifying an increase in measured heat transfer.

More recently, the effects of free-stream vorticity normal to a three-dimensional flat plate were investigated by Goldstein, Leib and Cowley. Their efforts addressed the strengthening of the vorticity due to the effect of the plate leading-edge bluntness and a resulting change in the spanwise flow. This flow modification changes the boundary-layer profile, which results in a reduction of the wall shear stress. The shear stress eventually goes to zero and the boundary-layer separates. Their observations support those for experimental studies of turbulent boundary-layer separation.

Pop and Gorla solved the second-order boundary-layer equations for an incompressible non-Newtonian flowfield over a continuously moving semi-infinite flat plate. They developed the equations following Van Dyke’s method of matched asymptotic expansions and numerically solved for the velocity distribution and skin friction for various non-Newtonian fluids. They observed a higher velocity distribution for pseudoplastic fluids than for dilatant fluids. Also, for pseudoplastic fluids the skin friction coefficient decreases asymptotically while dilatant fluids exhibit an asymptotic increase in skin friction.

Sawley, and Wuthrich compared the numerical solutions of the first and second-order boundary-layer equations with the Euler equations for hypersonic flow in chemical non-equilibrium over a double ellipse geometry, modeling a re-entry vehicle. For the pressure and tangential velocity profiles they showed marked improvement with the second-order equations relative to the Euler solutions. Their work identified an increase in Stanton number and skin friction coefficient for the second-order results. These conclusions are consistent with the observations of Hayes and Probstein.
that with a nonuniform flowfield where the local velocity increases with increasing $y$ (normal to plate surface), numerical evaluation of ensuing vorticity will show a consequential increase in the skin friction coefficient and Stanton number.$^{25}$

These examples are by no means meant as an exhaustive listing of the efforts to date. However, they present the significant conclusions and expectations for second-order, compressible boundary-layer flows.
APPLICATION OF CFD

Description of CFD Solver (GASP)

The use of Computational Fluid Dynamics (CFD) for the solution of complex flow problems provides previously unobtainable insight to the behavior of fluid flow. CFD is a critical tool for expanding our understanding of the physics of fluid dynamics.

Since the solution of the complete Navier-Stokes equations is possible using CFD, we can solve for the physical response of the flow to an external gradient in the flow properties. Once the CFD solution is found, the same problem can then be solved using second-order boundary-layer theory. Comparison of the CFD solution with the results obtained from second-order boundary-layer theory will indicate the accuracy of the second-order theory in characterizing the effects of the flow gradients on the boundary-layer.

The CFD analysis will be performed using the General Aerodynamic Simulation Program (GASP) developed by AeroSoft, Inc. The capabilities of GASP are described by AeroSoft as follows:

GASP solves the integral form of the time-dependent, three-dimensional Reynolds-Averaged Navier-Stokes (RANS) equations subject to boundary and initial conditions. It is also capable of solving subsets of the RANS equations including two-dimensional and axi-symmetric problems, the Thin-Layer Navier-Stokes (TLNS) equations, the Parabolized Navier-Stokes (PNS) equations, and the Euler equations. GASP is a fully conservative shock capturing CFD code because of the consistent manner in which properties such as volume, surface area, direction cosines, and numerical flux functions are evaluated. Additionally, GASP can solve both laminar and turbulent flow problems.

The GASP flow solver is based on a finite volume analysis of the governing Navier Stokes equations. Since the solver is finite-volume versus finite-difference based, GASP places the computational grid nodes in the center of the individual control volumes or cells, which are defined with i,j,k grid indices. The computations are therefore performed such that the flow state is stored in
the cell centers. All flux terms are handled by interpolating the flow state from the cell center to the relevant cell face.

Boundary condition information in GASP is also stored in cell centers. Boundary data is evaluated over the cell-centered dimensions, \((-1, \text{idim}+1, -1, \text{jdim}+1, -1, \text{kdim}+1)\), where \text{idim}, \text{jdim}, \text{kdim} are the maximum \text{i,j,k} values respectively. At any non-solid boundary, GASP evaluates the boundary conditions using two boundary "ghost" cells, located on the exterior of the boundary. However, for solid wall boundaries, the boundary cells are located on the boundary face. Figure 1 is an example of the treatment of boundary conditions in GASP. The \((i=1,j=1,k=1)\) cell is pictured with two non-solid boundaries along the \text{i}, \text{k} and \text{j}, \text{k} planes. The boundary conditions on these planes are therefore evaluated with the ghost boundary cells shown. The \text{i,j} plane is shown as a solid wall boundary, requiring the boundary condition be evaluated on that cell face, \((i=1,j=1,k=0)\).²⁷

![Figure 1. Treatment of Boundary Cells in GASP.](image)

The GASP computational theory starts with the integral form of the full three-dimensional Reynolds-Averaged Navier-Stokes equations;

\[
\frac{\partial}{\partial t} \iiint \nu dV + \oint_A (\vec{F} - \vec{F}_\nu) \cdot \hat{n} dA = \iint \Phi dV
\]

Equ. (30)
where $\tilde{F}$ is the inviscid flux vector and $\tilde{F}_v$ is the viscous flux vector, $V$ is the cell volume, $\hat{n}$ is the unit vector normal to the appropriate cell face, and $Q$ and $\Phi$ are defined below. Next the inviscid and viscous flux term, $\oint_{\partial A} (\tilde{F} - \tilde{F}_v)$, is approximated as the sum of the inviscid and viscous flux terms across each face of the given control volume. Equation (2) can thus be rewritten as:

$$\frac{\partial \langle Q \rangle}{\partial t} + \frac{\partial \langle q \rangle}{\partial t} V + \sum_{j=1}^{\text{face}} (\tilde{F} - \tilde{F}_v) \cdot \hat{n} \Delta A = \langle \Phi \rangle V$$

Equ. (31)

with $\langle Q \rangle = \frac{1}{V} \iiint_{\nu} Q(x,y,z,t) dV$ and $\langle \Phi \rangle = \frac{1}{V} \iiint_{\nu} \Phi(x,y,z,t) dV$

and where $Q$ and $q$ are defined as the state vector of conserved and primitive flow variables, respectively, which for $q$ is given as:

$$q = \begin{bmatrix} \rho_l \\ u \\ v \\ w \\ e_{ni} \\ p \\ K \\ e_t \end{bmatrix}$$

Equ. (32)

where $\rho_l$ represents the fluid density for each individual species in Chemistry flow solutions, and $e_j$ represents the internal energy per unit mass of each species. $K$ is the turbulent kinetic energy and $\varepsilon_t$ is the dissipation rate of turbulent kinetic energy.

The $\Phi$ term in Equation (31) is the source term vector used for modeling turbulent and chemically reacting flows, given as;
In CFD, the discretized equations can be solved in either space or time. For parabolic or hyperbolic problems space solutions are performed by evaluating the cell properties for grid planes normal to the marching direction (usually the dominant flow direction), one plane at a time. The solution of downstream planes is thus wholly dependent on the values determined for the upstream planes. For elliptic problems time marched solutions are performed by computing the state of the entire computational domain for a given “slice” of time, then incrementing the governing equations by a small time step and evaluating the new time “slice” using the previous time step solution. The solution is continually marched through time until the flow reaches the equilibrium state within the accuracy of the selected convergence criteria. The marching scheme employed determines how Equation (30) is approximated for the numerical analysis.

**Spatial Discretization of Equation (30)**

For computational solutions marched in space, Equation (30) is rewritten as:

$$\frac{\partial Q}{\partial q} \frac{\partial q}{\partial t} V + R(q) = 0$$

*Equ. (34)*

where $R(q)$ is a residual grouping of all the flux terms. Using the i,j,k coordinate system standard to GASP finite volume analysis we can expand the residual term in Equation (34) to:
\[ R(q) = \sum_{i=1}^{n_{\text{face}}} \left( \hat{F} - \hat{F}_o \right) \cdot \hat{n} \Delta A - \Phi V = \left( \hat{F} - \hat{F}_o \right)_{i+j+k} + \left( \hat{F} - \hat{F}_o \right)_{i+j+k} - \left( \hat{F} - \hat{F}_o \right)_{i+j+k} \]

\[ \text{Equ. (35)} \]

where the i,j,k indices represent cell centers and the 1/2 spatial step represents the appropriate cell face. Computation of both the inviscid and viscous flux terms requires interpolation of the flow primitive variables to the appropriate cell face for the given computation. The selected interpolation method controls the spatial accuracy of the solution. The standard interpolation for the q solution vector is represented by;

\[ (q_L)_{i+j} = q_i + \frac{\phi}{4} \left( (1-\kappa) \nabla + (1+\kappa) \Delta \right) q_i \]

\[ (q_R)_{i+j} = q_i - \frac{\phi}{4} \left( (1+\kappa) \nabla + (1-\kappa) \Delta \right) q_i \]

\[ \text{Equ. (36)} \]

with

\[ \Delta q_i = q_{i+1} - q_i \]

\[ \nabla q_i = q_i - q_{i-1} \]

\[ \text{Equ. (37)} \]

for interior and ghost boundary (non-solid boundary) cells, while for solid boundaries

\[ \Delta q_i = 2(q_{i+1} - q_i) \]

\[ \nabla q_i = 2(q_i - q_{i-1}) \]

\[ \text{Equ. (38)} \]

where the value of \( \kappa \) controls the spatial accuracy of the interpolation as selected in the GASP input file, and \( \phi \) is zero for first-order accuracy, and one for higher-order accuracy.

For numerical modeling of the second-order boundary-layer problem, the inviscid flux calculations in the normal, \( y \), direction were performed with third order upwind biased accuracy (\( \kappa=1/3 \) in Equation (36)) using the Roe split flux model\(^{29}\). The Roe split flux model solves the Riemann conditions which develop due to the discontinuities at cell interfaces. The contributions from multiple characteristic waves are summed over the appropriate cell interfaces to arrive at an
GASP allows limiting of the higher-order correction term, on the right side of Equation (36), to maintain stability and reduce oscillations in regions of large flow gradients. The Min-Mod limiting model was applied to the primitive variables for the inviscid Roe split flux calculations in the \( y \) direction. The limiting when applied to Equation (36) gives;

\[
(q_L)_{i+\frac{1}{2}} = q_i + \frac{1}{4} \left[ (1 - \kappa) \overline{\nabla} + (1 + \kappa) \overline{\Delta} \right] q_i, \quad \text{Equ. (39)}
\]

\[
(q_R)_{i-\frac{1}{2}} = q_i - \frac{1}{4} \left[ (1 + \kappa) \overline{\nabla} + (1 - \kappa) \overline{\Delta} \right] q_i,
\]

where;

\[
\overline{\Delta} = \min \text{mod}(\Delta, \beta \nabla)
\]

\[
\overline{\nabla} = \min \text{mod}(\nabla, \beta \Delta)
\]

\[
\text{min mod}(x, y) = \text{sign}(x) \max(0, \min[x(\text{sign}(y)), y(\text{sign}(x))])
\]

and with \( \beta \) given by;

\[
\beta = \frac{3 - \kappa}{1 - \kappa}, \quad \text{Equ. (41)}
\]

The inviscid flux in the streamwise, \( x \), direction was determined with a second-order, fully upwind accurate (\( \kappa=1 \) in Equation (36)) full flux calculation. The full flux calculation determines the inviscid flux vector, \( \vec{F} \) normal to the cell face using information from only the left state for the given flux direction.

For the viscous flux terms the flowfield characteristic variables are interpolated to the cell faces through basic averaging techniques. The viscous flux function is expressed in local Cartesian coordinates and transformed into a generalized curvilinear frame of reference. Viscous effects can be limited to the local coordinate direction only, with no contribution from the neighboring directions. This reduces the solution to the Thin-Layer Navier-Stokes (TLNS) equations. Addition of the cross-directional viscous derivatives introduces the full Navier-Stokes correction to the given coordinate direction. Therefore, the viscous terms can be controlled independently for each direction.
The current modeling uses Thin-Layer plus cross derivative terms for computing the viscous flux vector $\vec{F}_v$ in the $y$ direction, where the viscosity will change most drastically. No viscous terms are included for the $x$-direction. Therefore the full Navier-Stokes correction is taken only in the normal flow direction.

The coefficient of laminar viscosity is determined from empirical curve-fits, of which GASP supports six models. The curve-fit based on Sutherland’s Law for a Perfect Gas is used for this problem. The laminar viscosity is thus determined as;

$$\mu_i = T^{4.5} \left( \frac{E_i}{T + F_i} \right), \quad i = 1...N$$

Equ. (42)

where $E_i$ and $F_i$ are empirical coefficients.

The coefficient of laminar thermal conductivity is also computed from a curve fit, selected from four available models. Sutherland’s model is also applied here, with different values for the $E_i$ and $F_i$ coefficients. Laminar mass diffusion is modeled using the Stefan-Maxwell equation, Fick’s law of diffusion, with diffusion coefficients calculated using a constant Schmidt number.\textsuperscript{31}

When space marching viscous flow problems, the inviscid flux function for the marching direction must be full flux. With this setting, GASP identifies the case as viscous space marching and applies the Vigneron technique. Such solutions are typically performed only for supersonic flows, where the majority of the flowfield characteristic waves propagate downstream. The boundary-layer region will however produce an inviscid, upstream traveling, negative characteristic wave. The Vigneron technique modifies the pressure term of the full inviscid flux, producing a positive, downstream characteristic wave, thus reducing the elliptical TLNS equations to the Parabolized Navier-Stokes (PNS) equations. This technique is employed for all space marched solutions in this study.

**Time Discretization of Equation (30)**

Starting from Equ. (30) as was done for Spatial Discretization, we can expand the residual, $R(q)$, in terms of $i,j,k$ indices for GASP time integration;
\[
R(q) = \sum_{i=1}^{n_{face}} (\mathbf{F_i} - \mathbf{F}_{i-1}) \cdot \mathbf{\hat{n}} \Delta A - \mathbf{\Phi} \mathbf{V} = (\mathbf{\hat{F}_i} - \mathbf{\hat{F}_{i-1}}) + \\
(\mathbf{\hat{G}_i} - \mathbf{\hat{G}_{i-1}}) + \Phi_{i,j,k} \mathbf{V}_{i,j,k}
\]

Equ. (43)

where \( F, G, \) and \( H \) vectors represent fluxes in the \( i,j,k \) directions respectively. GASP supports both explicit and implicit time integration schemes. For the current numerical analysis the time marched solutions use implicit time integration. GASP applies the Euler-Implicit method, which is first-order accurate in time;

\[
\left[ \frac{\partial Q \mathbf{V}}{\partial t} + \frac{\partial}{\partial q} R(q^n) \right] \Delta q = -R(q^n)
\]

Equ. (44)

where the second term is a symbolic operator defined as;

\[
\frac{\partial R}{\partial q} \Delta q = \left( \frac{\partial (\mathbf{\hat{F}_i} - \mathbf{\hat{F}_{i-1}})}{\partial q} \right) + \\
\left( \frac{\partial (\mathbf{\hat{G}_i} - \mathbf{\hat{G}_{i-1}})}{\partial q} \right) + \frac{\partial}{\partial q} \mathbf{\Phi} \Delta q
\]

Equ. (45)

and the inviscid contributions are simplified with a first-order approach, as;
\[
\begin{align*}
\left( \frac{\partial \hat{F}}{\partial q} \right)_{i+\frac{1}{2},+\frac{1}{2}} &= \left( \frac{\partial \hat{F}}{\partial q_L} \right)_{i+\frac{1}{2},+\frac{1}{2}} \Delta q_i + \left( \frac{\partial \hat{F}}{\partial q_R} \right)_{i+\frac{1}{2},+\frac{1}{2}} \Delta q_{i+1} \quad \text{Equ. (46a)} \\
\left( \frac{\partial \hat{F}}{\partial q} \right)_{i-\frac{1}{2},-\frac{1}{2}} &= \left( \frac{\partial \hat{F}}{\partial q_L} \right)_{i-\frac{1}{2},-\frac{1}{2}} \Delta q_{i-1} + \left( \frac{\partial \hat{F}}{\partial q_R} \right)_{i-\frac{1}{2},-\frac{1}{2}} \Delta q_i \\
\end{align*}
\]

while the viscous flux terms are approximated with only the Thin-Layer contributions in the flux direction and no cross-derivative terms as;

\[
\begin{align*}
\left( \frac{\partial \hat{F}_v}{\partial q} \right)_{i+\frac{1}{2},+\frac{1}{2}} &= \left( \frac{\partial \hat{F}_v}{\partial q_L} \right)_{i+\frac{1}{2},+\frac{1}{2}} \Delta q_i + \left( \frac{\partial \hat{F}_v}{\partial q_R} \right)_{i+\frac{1}{2},+\frac{1}{2}} \Delta q_{i+1} \quad \text{Equ. (47a)} \\
\left( \frac{\partial \hat{F}_v}{\partial q} \right)_{i-\frac{1}{2},-\frac{1}{2}} &= \left( \frac{\partial \hat{F}_v}{\partial q_L} \right)_{i-\frac{1}{2},-\frac{1}{2}} \Delta q_{i-1} + \left( \frac{\partial \hat{F}_v}{\partial q_R} \right)_{i-\frac{1}{2},-\frac{1}{2}} \Delta q_i \\
\end{align*}
\]

In Equations (46) and (47) the time increments, \( \Delta q \) terms, are evaluated using two-factor Approximate Factorization\(^{32}\), reducing the problem to a set of two block-tridiagonal systems of equations.

**Physical Modeling in GASP**

Modeling flow problems for computational analysis with GASP requires mapping the physical space to a computational space which is defined with a binary grid file. This grid file is developed prior to execution of the GASP analysis routine. Development of the grid file is independent of GASP and requires only that the data be properly formatted for GASP execution. GASP is equipped with several FORTRAN binary file generation routines which convert \( x, y, z \) coordinate data to the appropriate GASP grid input format. These routines are used for the computational grid designs in this study.

Since the problem of interest involves a two-dimensional shape with no curvature we can easily apply a rectangular grid as the computational space, with the \( y \)-direction associated with the \( i \) grid index and the \( x \)-direction associated with the \( k \) grid index. The maximum \( i \) and \( k \) grid lines are specified as \( idim \) and \( kdim \) respectively. The leftmost boundary, near the plate leading edge, is
referred to as $k_0$, and the computational boundary at the plate surface is referred to as $i_0$. Figure 2 is a graphical representation of the basic rectangular grid design.

![Figure 2. Physical Model Mapping to Computational Space.](image)

GASP execution minimally requires two other input files; a control file and a zone file. The primary GASP control file contains freestream flow variables, computational zone names and a description of the marching scheme and related tolerances. GASP can solve over multiple grid zones with varying marching schemes. For each GASP computational zone a zone input file is also required. The zone file contains information on the boundary conditions, inviscid and viscous fluxes and the chemistry and thermodynamics models for the given flowfield.

For the current study all CFD analysis was performed on a Sun SparcStation2 with GASP version 2.2.10 running SunOS 4.1.3, Openwindows 3.0, with 64 MB RAM. GASP was compiled on the same platform with Sun FORTRAN 1.4.

**GASP Post-Processing ("PRINT")**

Analysis of GASP computational solutions is simplified through the use of the "PRINT" post-processing routine. "PRINT" is capable of processing data for approximately 80 flow variables. Data output can be formatted as line output, tab delineated output to a table file, various PLOT3D output formats, and TECPLOT format. A "PRINT" input file specifies the desired output format, variables of interest, and the section of computational space for which output is desired.
For this study, the TECPLOT interactive data visualization software from AMTEC Engineering provides the capability for graphical analysis of the GASP output. TECPLOT is a popular plotting tool for analysis of CFD and structural Finite-Element Analysis data.33

Analysis Parameters

Several flow parameters will provide useful insight to the boundary-layer growth and characteristics. The flowfield velocity components will provide an indication of the initial development and continual growth of the boundary-layer along the length of the plate. Of particular interest is the tangential velocity component, $u$. Contour plots of the variation in the $u$ velocity component as a function of the grid geometry indicate the boundary-layer growth and interaction with the external flow region. The boundary-layer thickness, shape factor, displacement thickness, and momentum thickness serve as useful descriptors of the flow characteristics and boundary-layer development.

The GASP output parameters relevant to this analysis include the $x$ and $y$ coordinates, $u$ and $v$ velocity components, Mach number, skin friction coefficient ($c_f$), and nondimensional heat transfer ($St$). From these flow parameters it is possible to calculate the remaining boundary-layer parameters, as discussed below.

Boundary-layer Thickness, $\delta$

The boundary-layer thickness, $\delta$, is a measure of the height of the boundary-layer. The flow above this height is considered to be external flow. We define the boundary-layer thickness as the height above the flat plate where the local velocity is equal to 99% of the freestream velocity, $U_{\infty}$. This definition serves to quantify the strength of the boundary-layer at a given point along the plate.

The boundary-layer thickness is determined by post-processing the $u$ velocity parameter as output from GASP. At each $x_0$ location we evaluate the variation in $u$ normal to the plate, applying polynomial interpolation to locate $u(x_0, \delta) = 0.99 U_{\infty}$. Varying $x_0$ from the leading edge to the trailing edge gives $\delta(x)$. 
For the cases with nonuniform external flow, $U_\infty$ is the velocity at the boundary-layer edge, $U_e$, which we define as $U_\infty$ for the uniform flow case. This definition provides consistency for comparison of the nonuniform flow cases with the uniform solution. Comparisons of the boundary-layer parameters for nonuniform solutions against the uniform solution will directly indicate the flow response to the given gradient.

**Displacement Thickness, $\delta^*$**

The boundary-layer develops as an inner region of slower moving fluid. As required by the law of Conservation of Mass, as this slower region of fluid flow grows, it gradually displaces the outer region of free-stream flow. The distance the external flow is displaced by the boundary-layer is termed the displacement thickness, noted as $\delta^*$. The boundary-layer displacement thickness is defined as:

$$\delta^* = \int_0^\delta \left(1 - \frac{\rho u}{\rho_\infty U_\infty}\right) dy$$

Equation (48)

A numerical integration technique is used to determine the displacement thickness at a given $x$ location along the plate. The numerical integration algorithms used in this analysis were taken from Press, Teukolsky, Vetterling, and Flannery\textsuperscript{34}. The methodology relies upon application of the trapezoidal rule.

**Momentum thickness, $\theta$**

The momentum thickness, $\theta$, further quantifies the development of the boundary-layer. Momentum thickness is a measure of the flow momentum loss due to wall shear stress.

$$\theta = \int_0^\delta \frac{\rho u}{\rho_\infty U_\infty} \left(1 - \frac{u}{U_\infty}\right) dy$$

Equation (49)

Numerical integration provides a means for determining the momentum thickness at any $x$ location. The trapezoidal rule is applied in the same manner as for the displacement thickness.
**Shape Factor, H**

The shape factor provides a quantitative measurement of the state of the boundary-layer. The value of the shape factor indicates if the boundary-layer is a stable, laminar flow or, alternatively, if the flow is approaching separation and turbulence. It is evaluated as the ratio of displacement thickness to momentum thickness;

\[
H = \frac{\delta^*}{\theta}
\]

Equ. (50)

For the case of flat plate stable laminar flow the shape factor is 2.59. Therefore, the numerical results should provide a shape factor of approximately 2.59 for the Case1, baseline flow.

**Stanton Number (St)**

The Stanton number is a nondimensional form of the heat transfer, \( h \). It is defined as;

\[
St = \frac{h}{(\rho Uc_p)_{\infty}}
\]

Equ. (51)

where \( c_p \) is the specific heat constant pressure, which for air is, \( c_p = 1005 \text{ m}^2/\text{s}^2\text{K} \).

The GASP post-processing program directly outputs the Stanton Number.

**CFD solutions to study effects**

To evaluate the validity of second-order boundary-layer theory it is desirable to solve several different flow cases with CFD for later comparison against theoretical results. The use of multiple solution cases provides a broader understanding of the accuracy and effectiveness of the theory. The application of the second-order theory to several cases will quantify its limitations and allow for an assessment of its applicability to other real-world engineering geometries.

The following section details the three primary flow cases solved for this analysis. The study includes a baseline, uniform flowfield case (Case1), a flowfield case including a linear stagnation enthalpy gradient (Case2), and a flowfield case with a nonlinear stagnation enthalpy gradient (Case3). For the nonuniform flowfields several gradient strengths were investigated to study the balance of second-order response and computational resolution.
FLOWFIELD CASES - DESCRIPTION

Physical Geometry

To simplify the modeling efforts involved in the CFD analysis all of the cases studied assume the same physical geometry. The supersonic flow over a two-dimensional flat plate of length 1.0 meter is used for all cases. The flat plate is a familiar analysis model and its simplicity reduces the effort needed to solve the second-order boundary-layer equations for future comparisons between the theory and the CFD results. As mentioned previously, Emanuel provides a solution of the second-order boundary-layer equations for the two-dimensional, flat plate geometry. It is therefore desirable to maintain a geometry consistent with Emanuel’s solution to provide a direct comparison of CFD and theory.

We assume no velocity changes in the spanwise direction with the plate effectively of infinite span. The coordinate system is right-handed with x associated downstream, y normal to the plate surface and the origin at the plate leading edge. Even though we solve the flow from the leading edge to trailing edge, we only use the region between 0.2 (m) to 0.6 (m) for this study. This is to avoid the effect of the boundaries. The study region allows for substantial boundary-layer growth and provides a region of flow analysis where second-order effects can be clearly exposed and studied.

The baseline freestream flow properties for air as a perfect gas are denoted with the $\infty$ subscript and listed in Table 2.
Table 2. Baseline Freestream Flow Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach Number ($M_\infty$)</td>
<td>3.0</td>
</tr>
<tr>
<td>y Velocity Component ($V_\infty$)</td>
<td>0.0 m/s</td>
</tr>
<tr>
<td>x Velocity Component ($U_\infty$)</td>
<td>1041.385 m/s</td>
</tr>
<tr>
<td>Pressure ($P_\infty$)</td>
<td>1.01304x10^5 Pa</td>
</tr>
<tr>
<td>Temperature ($T_\infty$)</td>
<td>300K</td>
</tr>
<tr>
<td>Density ($\rho_\infty$)</td>
<td>1.177 kg/m^3</td>
</tr>
<tr>
<td>Viscosity Coefficient ($\mu_\infty$)</td>
<td>1.84629x10^{-5} Ns/m^2</td>
</tr>
<tr>
<td>Ratio of Specific Heats ($\gamma$)</td>
<td>1.4</td>
</tr>
<tr>
<td>Specific Heat Constant Pressure ($C_p$)</td>
<td>1005 m^2/(s^2K)</td>
</tr>
<tr>
<td>Prandtl Number (Pr)</td>
<td>1.0</td>
</tr>
</tbody>
</table>

The baseline conditions assume a uniform flowfield at the plate leading edge ($x=0.0$ m). The plate surface is assumed to have a constant temperature of $T_{wall}=300$K. Figure 3 graphically depicts the physical geometry for the baseline flow.

![Figure 3. Initial Baseline Physical Geometry.](image)

Estimation of the boundary-layer thickness at trailing edge ($x=1.0$ meter) will provide guidance for the selection of the computational space. The boundary-layer thickness for a laminar flow can be estimated as;
Given the baseline flow conditions described in Table 2, the local Reynolds number, \( \text{Re}_x \), is evaluated as:

\[
\text{Re}_x = \frac{\rho V L}{\mu} = \frac{(1177 \text{ kg/m}^3)(1041.385 \text{ m/s})(1.0 \text{ m})}{1.84629 \times 10^{-5} \text{ Ns/m}^2} = 6.63878 \times 10^7 \quad \text{Equ. (53)}
\]

which gives a boundary-layer thickness at \( x = 1.0 \) (m) of,

\[
\delta = (1.0) \left( 6.63878 \times 10^7 \right)^{-\frac{1}{2}} = 6.137 \times 10^{-4} \text{ m} \quad \text{Equ. (54)}
\]

**Baseline Computational Case (Case I)**

Solution Case I provides the baseline computational results for comparison with the gradient flow cases. Case I is characterized by a constant uniform, irrotational Mach 3.0 flowfield at the plate leading edge. Figure 4 depicts the physical geometry for the baseline, Case I, computations.

![Figure 4. Case I Physical Geometry.](image)

The boundary conditions for the Case I computations are given in Table 3.
Table 3. Case 1 Boundary Conditions Defined

<table>
<thead>
<tr>
<th>Grid Boundary</th>
<th>Boundary Condition</th>
<th>GASP Input Variables</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k_0$ (left)</td>
<td>Fixed at freestream</td>
<td>$k_0bc= -1$</td>
</tr>
<tr>
<td>$k_{dim}$ (right)</td>
<td>First-order from interior cells</td>
<td>$k_{dim}bc= -3$</td>
</tr>
<tr>
<td>$i_0$ (plate)</td>
<td>No-slip, Constant temperature wall</td>
<td>$i_0bc= +10, twall=300K$</td>
</tr>
<tr>
<td>$i_{dim}$ (top)</td>
<td>Subsonic inflow/outflow</td>
<td>$i_{dim}bc= -5$</td>
</tr>
</tbody>
</table>

The GASP boundary condition input variables in Table 3 refer to the appropriate grid boundary as shown in Figure 4. The +/- sign of the boundary conditions denotes a split flux or full flux condition, respectively. The split flux specification instructs GASP to evaluate the flow values at the given boundary with one state determined from the computed boundary values as specified through the boundary condition and the other state interpolated from the interior cell values. The full flux boundary values are determined using the computed values from given boundary condition for both flux states.\(^{35}\)

**Linear Stagnation Enthalpy Functions (Case 2)**

The second computational case, Case 2, adds a linear stagnation enthalpy gradient to the freestream flow, prior to the plate leading edge, as shown in Figure 5.
The stagnation enthalpy gradient cannot be directly input as the \( k_0 \) boundary condition in GASP. However, GASP will support pointwise input of flow velocity and pressure variables at the boundary. As discussed earlier, the Substitution Principle is applied to characterize the proper flowfield.

Using the Substitution Principle, we assume the following form for \( \lambda \):

\[
\lambda = 1 + h'_0 y
\]

Equ. (55)

where \( h'_0 \) is the stagnation enthalpy gradient at the plate, \( \frac{dh_0}{dy}_{wall} \), and \( y \) is the distance measured normal to the plate surface.

Equations (27) provide the transformed solution for a parallel flow, and indicate that the transformed pressure equals the baseline flow pressure, while the transformed velocity is determined from:

\[
u = \lambda \frac{\partial}{\partial y} U_{\infty}
\]

Equ. (56)

Equation (46) provides a velocity gradient consistent with the linear stagnation enthalpy gradient we desire. The transformed velocity gradient and pressure are then provided as the pointwise \( k_0 \) boundary condition for GASP. Modifying the gradient strength will provide flexibility in controlling the nonuniformities.

Substituting Equations (45) into Equation (46) for \( \lambda \), we get the transformed velocity as,

\[
u = \left(1 + h'_0 y\right) \frac{\partial}{\partial y} U_{\infty}
\]

Equ. (57)

Therefore we transform the uniform flowfield of Case1, where \( U_{\infty} \) is known, using Equation (57). By varying the choice of \( h'_0 \) we select several gradient strengths and determine the appropriate initial velocity profile for the flowfield.

The boundary conditions for the Case2 computations are provided in Table 4.
Table 4. Case2 Boundary Conditions Defined

<table>
<thead>
<tr>
<th>Grid Boundary</th>
<th>Boundary Condition</th>
<th>GASP Input Variables</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k_0$ boundary</td>
<td>Read from bcqfile</td>
<td>$k_0bc = -2$</td>
</tr>
<tr>
<td>$kd$ boundary</td>
<td>First-order from interior cells</td>
<td>$kdimbc = -3$</td>
</tr>
<tr>
<td>$i_0$ boundary</td>
<td>No-slip, Constant temperature wall</td>
<td>$i_0bc = +10$, twall=300K</td>
</tr>
<tr>
<td>$id$ boundary</td>
<td>Subsonic in/outflow (Riemann)</td>
<td>$idimbc = -5$</td>
</tr>
</tbody>
</table>

Note that the $k_0$ boundary condition, "read from bcqfile" instructs GASP to read a file of pointwise flow quantities for that boundary.

**Nonlinear Stagnation Enthalpy Functions (Case 3)**

The third case introduces a nonlinear stagnation enthalpy gradient in the initial flowfield.

Figure 6 is a representation of the physical geometry for Case3. However, the nonlinear gradient shown is not necessarily the shape of the gradient actually used in the computational analysis.

![Figure 6. Case3 Physical Geometry.](image)

As with Case2, we specify a stagnation enthalpy gradient and, applying the Substitution Principle, transform the baseline flowfield to a new flowfield consistent with the gradient choice. Here we assume the $\lambda$ parameter to be of the form;
\[ \lambda = \left[ 1 + \sigma \ erf (3.5y) \right]^{-2} \]  
Equ. (58)

where the \( \sigma \) parameter controls the shape of the nonlinear gradient and \( erf \) is the error function, defined as,

\[ erf(x) = \frac{2}{\sqrt{\pi}} \int_0^x e^{-t^2} dt \quad \text{for all } x \]  
Equ. (59)

The \( \lambda \) function selected in Equation (58) has been successfully used in association with the Substitution Principle for compressible flow applications. Therefore, following the procedure outlined for Case2, we arrive at the following relation for the transformed velocity,

\[ u = \left[ \left( 1 + \sigma \ erf (3.5y) \right)^{-2} \right]^{1/2} U_\infty \]  
Equ. (60)

The boundary conditions used in the Case3 computational runs are shown in Table 5. They are consistent with the Case2 boundary conditions in Table 4.

**Table 5. Case3 Boundary Conditions Defined**

<table>
<thead>
<tr>
<th>Grid Boundary</th>
<th>Boundary Condition</th>
<th>GASP Input Variables</th>
</tr>
</thead>
<tbody>
<tr>
<td>( k_0 ) boundary</td>
<td>Read from bcqfile</td>
<td>( k_{0bc} = -2 )</td>
</tr>
<tr>
<td>( kdim ) boundary</td>
<td>First-order from interior cells</td>
<td>( k_{dimbc} = -3 )</td>
</tr>
<tr>
<td>( i_0 ) boundary</td>
<td>No-slip, ( T = twall )</td>
<td>( i_{0bc} = +10, twall = 300K )</td>
</tr>
<tr>
<td>( idim ) boundary</td>
<td>Subsonic in/outflow(Riemann)</td>
<td>( id_{imbc} = -5 )</td>
</tr>
</tbody>
</table>
Computational Grid Development

Initial GASP solution runs on Cases 1 and 2 reveal several important considerations in the design of the computational grid. As the numerical experimentation progresses, insight is gained and the grid is gradually modified to provide increased accuracy and reasonable run times.

In the following discussion the grid characteristics are described in terms of the maximum i,k values. Therefore, all references to grid size are made using the convention ("idim", "kdim"). Also, each numerical exercise for a solution Case is referred to as a computational “run”. Included in the Appendix is the computational run log maintained during the numerical experimentation. Throughout this discussion, references are made to the Appendix using the notation (run #). The Appendix run log is ordered by run # and can serve as a useful guide to the text.

GASP supports solution of the Navier-Stokes equations using both time and space discretization methods. Therefore, in the discussion of computational runs, the distinction of solution method is made by denoting the run as either a space solution or a time solution. For this study, the space marched runs were setup to solve the PNS equations, by setting the inviscid flux for the k marching direction to full flux, allowing GASP to apply the Vigneron technique discussed earlier. The time marched solutions solve the TLNS approximation of the full N-S equations, applying only thin-layer cross derivatives in the non-marched (i and k) directions. The accuracy of the GASP run is controlled through specification of a convergence tolerance. For solutions marched in both space and time, the tolerances are specified as “space tolerance”/”time tolerance”.

Blasius Baseline Grid

The GASP CFD solver includes a sample case which solves the TLNS equations for a subsonic, uniform flowfield over a 1.0 meter long flat plate. This case essentially models the Blasius
exact solution for the first-order boundary-layer equations. For the solution, the computational space is defined with the (41x21) point grid show in Figure 7.

The “Blasius grid” serves as the starting point for the development of the computational space for the second-order effects analysis. GASP solution runs using the “Blasius grid” without modification are attempted to baseline the grid for further refinement.

![Blasius Grid](image)

**Figure 7. Blasius Grid.**

Preliminary GASP solution runs use nonuniform grid spacing in the $y$-direction and a uniform grid spacing in the $x$-direction, as shown for the Blasius grid in Figure 7. The nonuniform $y$ spacing compresses the grid near the plate surface, concentrating cells within the height of the boundary-layer. It is desirable to place many cells within the boundary-layer region since this area involves steep gradients in critical flow parameters and is computationally sensitive. The Blasius grid
is developed using an exponential stretching algorithm in the y-direction. The algorithm uses the parameter \( \beta \) to control the level of grid compression. Values for \( \beta \) are greater than one with increasing \( \beta \) denoting less compression. Typical values are \( \beta = 1.05 \) for laminar flows and \( \beta = 1.0005 \) for turbulent flows.\(^{37}\)

The Blasius grid stretching algorithm populates the \( x,y \) grid coordinates using the following scheme, where \( k \) is associated with the \( x \) direction and \( i \) is associated with the \( y \) direction.

\[
\sum_{i=1}^{\text{dim}} \sum_{k=1}^{\text{dim}} x_{k,i} = x_{k,\text{dim}} \frac{(\eta_y - 1)}{(i \text{ dim} - 1)} + x_{k,1} \frac{(i \text{ dim} - \eta_y)}{(i \text{ dim} - 1)}
\]

Equ. (61)

\[
\sum_{i=1}^{\text{dim}} \sum_{k=1}^{\text{dim}} y_{k,i} = y_{k,\text{dim}} \frac{(\eta_y - 1)}{(i \text{ dim} - 1)} + y_{k,1} \frac{(i \text{ dim} - \eta_y)}{(i \text{ dim} - 1)}
\]

where the parameters are defined as;

\[
\eta_y = i \text{ dim} - \eta_y (i \text{ dim} - 1)
\]

\[
\eta_y = \beta \frac{(e^a - 1)}{(e^a + 1)} \quad \text{Equ. (62)}
\]

\[
a = \log \frac{(\beta + 1.0) (i \text{ dim} - i)}{(\beta - 1.0) (i \text{ dim} - 1)}
\]

The Blasius grid uses a stretching parameter of \( \beta = 1.05 \), with the overall grid dimensions set to \( x_{\text{max}} = 1.0 \) (m) and \( y_{\text{max}} = 0.03 \) (m), where \( x_{\text{max}} \) and \( y_{\text{max}} \) refer to the maximum \( x \) and \( y \) grid values respectively.

The Case1 input files are setup for a space marched solution using the (41x21) size Blasius grid, with a convergence tolerance of \( 1 \times 10^{-7} \) (run 110). These baseline Case1 runs provide a coarse representation of the boundary-layer, in Figure 8, without sufficient resolution for quantitative analysis or comparison to gradient flow results. Preliminary examination of the \( u \) velocity contour plot indicates the solution is not accurately resolved.
The flowfield is also inaccurately represented at the leading edge of the plate. The leading edge boundary condition for Case I is defined as a constant, uniform, freestream flowfield. However, the flowfield output from GASP depicts a small region of nonuniform flow at the left edge of the computational space, close to the plate surface. This inconsistency is studied and a solution is presented later in the text.

As a result of this preliminary analysis the grid is further refined to provide enhanced boundary-layer resolution. The computational grid is modified to provide 61 total grid points normal to the plate at an increased level of compression, $\beta=1.0005$. The intent is to improve the resolution of the boundary-layer. The solution converges to a tolerance of $1 \times 10^{-8}$ (run 111). Close inspection of the resulting boundary-layer, shown in Figure 9, indicates distinct improvements in flow resolution.
It appears these results could provide a useful preliminary baseline solution for comparison with the gradient flow cases.

![Figure 9. Case1 u Velocity Contour Plot - (61x21) Size Grid.](image)

Further evidence of the improvements gained with the refined grid is observed in a direct comparison of the boundary-layer thickness for both solutions. Figure 10 indicates that the Blasius grid provides a solution with a coarse boundary-layer thickness, particularly within the first 0.3 meters. The refined grid provides a significant improvement in resolution for the boundary-layer thickness. However, both solutions contain the inconsistency at the k_i boundary discussed earlier.
GASP solution runs for Case 2 are next attempted on the (61x21) point grid with $\beta = 1.0005$, and $h' = 0.5$ (run 112). This selection for $h'$ produces the stagnation enthalpy gradient and associated velocity profile shown in Figure 11.
The intent is to compare results for Case1 and Case2 on the same (61x21) point grid to determine if the second-order effects are evident. This Case2 run is resolved to a solution tolerance of $1 \times 10^{-8}$. However, comparisons with the Case1 solution for the (61x21) refined grid indicate little difference in boundary-layer parameters, as shown in Figure 12.
Figure 12. Boundary-Layer Parameters Case1/Case2 \( h' = 0.5 \) - (61x21) Size Grid
The displacement and momentum thickness in Figure 12 do not account for compressibility effects since they are calculated using the incompressible form, given in Equation (63).

\[
\delta^* = \int_0^\delta \left(1 - \frac{u}{U_\infty}\right) dy \\
\theta = \int_0^\delta \frac{u}{U_\infty} \left(1 - \frac{u}{U_\infty}\right)
\]

Equ. (63)

The \( u \) velocity contour plots also indicate the boundary-layer development for the cases is essentially identical. The results thus far seem to indicate that the gradient produced by \( h'_o = 0.5 \), Figure 10, is too weak to expose second-order flow effects in the solution. Possibly the effects are too small to resist being “washed out” by the boundary-layer development. This conclusion is supported by Emanuel’s observation that the experimental signal-to-noise ratio is a problem for capturing higher-order flow effects. Therefore, these preliminary analysis computations reveal no second-order effects and require improvement.

**Refined Grid Initial Modifications**

The first steps to improving the solutions involve further modifications to the (61x21) size refined computational grid. The primary emphasis is to increase the quantity of grid cells near the plate leading edge to eliminate the inconsistent nonuniform flow results described above. The computation space is modified to reduce the computation time wasted on the outer flow region, which appears to be an excessive portion of the solution space. Therefore, \( y_{\text{max}} \) is reduced from \( y_{\text{max}} = 0.03 \) to \( y_{\text{max}} = 0.005 \), based on the \( u \) velocity contour plots shown in Figures 8 and 9. The grid is further modified to include (61x101) cells, to provide enhanced resolution of the overall numerical results. An exponential stretching algorithm, equivalent to that used in the normal direction, is employed in the streamwise, \( k \), grid direction to compress the grid cells near the plate leading edge. The intent is to provide improved mesh refinement at the leading edge to eliminate the inconsistent velocity gradient observed for Case 1 at the \( k_o \) boundary in Figure 9. The resulting computational grid is shown in Figure 13.
As a further modification, the computational runs are setup to execute in two distinct marching blocks. The first block is performed as a space marched solution, solving the PNS equations for the entire computational domain. The second block solves the computational domain in time as a TLNS run, using the results of the space marched solution as initial conditions. The intent is to ensure the flowfield is solved using the TLNS equations, neglecting fewer higher-order terms which could contribute to the second-order effects.

Case1 is resolved on this grid to a convergence tolerance of $1 \times 10^{-9}/1 \times 10^{-10}$ (run 8), while Case2 is solved to $1 \times 10^{-7}/1 \times 10^{-11}$, with $\dot{h}_\phi = 10.0$ (run 12). This gradient strength produces the stagnation enthalpy and velocity gradients shown in Figure 14.
Comparison of the boundary-layer parameters for both runs, shown in Figure 15, reveals only minor flow differences in boundary-layer thickness near the plate trailing edge. The displacement and momentum thickness are calculated using the incompressible form, given by Equation (63). There still are no indications of significant second-order flow effects in the Case2 results. It seems likely that the linear enthalpy gradient still is not strong enough to produce distinct second-order effects.
Figure 15. Boundary-Layer Parameters Case 1/Case 2 $h_g = 10.0$ - (61x101) Size Grid.
Case2 Runs with Stronger Gradients

The lack of second-order flow effects in the solutions for Case2 requires modifications of the flow gradients. The stagnation enthalpy gradient for Case2 is increased in strength from $h'_{0} = 10.0$ to $h'_{0} = 100.0$, for the same grid properties (run 17).

Figure 16. Enthalpy and Velocity Gradient with $h'_{0} = 100.0$ - (61x101) Size Grid ymax=0.005 (m).

The solution for this Case2 run is solved in space to a convergence criteria of $1 \times 10^{-7}$, using the same grid shown in Figure 13. The contour plot of $u$ depicts a distinctly different boundary-layer from Case1, Figure 8.
At this point it is of interest to converge this solution in time, employing the Thin-Layer Navier Stokes equations and thus provide reassurance of the results and a consistent comparison with the Case 1 time-space results. Several attempts are made to converge this solution for Case 2 in time to a tolerance of $1 \times 10^{-16}$ using the space solution of Figure 17 as the initial flow conditions. Each attempt results in approximately nine or ten iterations on the solution and then the residual starts increasing, indicating a divergent run. The solution residual never improves beyond $6 \times 10^{-15}$. It is likely that the desired convergence tolerance, $1 \times 10^{-16}$, is drastically strong. For time marched solutions the computational errors of each grid plane contribute to the overall error on any given time step. Therefore, time solutions will generally propagate greater computational error through each step than similar space solutions. The time solutions will exhibit greater overall computational "stiffness".
Runs are then attempted to solve the same Case2 problem \((h_0'=100.0)\) in time initially, without using the space solution as the initial flowfield. These runs appear to be converging slowly, however they do not reach the final convergence criteria, \(1 \times 10^{-7}\), within 16h:50m:09s of computation time (run 25), and are therefore terminated prior to complete convergence.

Modifying the Courant-Friedrichs-Lewy (CFL) stability condition, Equation (64), is a possible solution to reducing the computational time experienced in the previous run. To improve the convergence computation time the CFL condition is relaxed from 10 to 20 for the time marched portion of the run. The run is setup to perform space marching to a tolerance to \(1 \times 10^{-7}\) followed by time marching to a convergence tolerance of \(1 \times 10^{-9}\). The solution completely converges in 8h:54mm:24s, indicating a 47% reduction in computation time (run 26). A more detailed explanation of the CFL condition and its influence is presented in the section titled “Aspect Ratio Effects”.
Figure 18. Boundary-Layer Parameters Case2 Space/Case2 Time $h_o = 100.0 - (61 \times 101)$ Size Grid
Figure 18 is a comparison of the boundary-layer parameters for the Case2 solutions using space marching and time marching. Note the difference in boundary-layer thickness and displacement thickness observable from approximately $x=0.5$ meters to the trailing edge. The slight increase in these parameters is the effect of the improved tolerance criteria for the time marched case, resulting in a more accurately resolved solution. However, it is not apparent that any second-order effects are captured using the time marched, TLNS solution that are not evident with the space marched, PNS results. Note that the displacement and momentum thickness are calculated without compressibility effects, using the incompressible form given by Equation (63).

A direct comparison of the boundary-layer parameters for the Case1 and Case2 space-time marched solutions is presented in Figure 19. There is a noticeable increase in the boundary-layer height for Case2. This effect was not observed in Figure 3 for $h_0 = 10.0$. 
Figure 19. Boundary-Layer Parameters Case1/Case2 Space-Time $h_0 = 100.0$ - (61x101) Size Grid.
Apparently the increase in gradient strength intensifies the effect. The increase is likely an indication of the influence of vorticity, accelerating the flow within the boundary-layer. The displacement and momentum thickness are calculated using the incompressible form given in Equation (63). Both parameters show only a slight variation from the baseline flow conditions. Therefore, with the (61x101) grid and the current linear gradient, \( h_0 = 100.0 \) the flowfield starts to exhibit the influence of small second-order effects in the boundary-layer thickness.

Examination of the contour plots for both Case1 and Case2 runs up to this point indicates a large, computationally expensive region of outer, freestream flow. Therefore, reduction of the computational space could likely improve the computation time and flow resolution while also exposing stronger second-order effects.

**Grid Standardization Efforts**

Given the wide range of grid sizes and convergence tolerances used for Case1 and Case2 in the previous grid development runs it is difficult to compare the results of Case2 against the baseline flow. Therefore it is desirable to develop a standardized grid based upon the knowledge gained to date.

The (61x101) size grid with \( \gamma_{\text{max}} = 0.001 \text{m} \) is used as the standard, based on the successful data obtained in Figures (15) and (19). New computational runs are attempted on this grid for Case1 and Case2 to provide standard solutions for direct comparison.

Case1 is run as a space solution and converged in 1h:33m:24s (run 47) to a tolerance of \( 1 \times 10^{-8} \). The \( u \) velocity contour plot, Figure 20, reveals a similar but larger boundary-layer shape than the results shown for Case1 in Figure 9.
Case 2 is solved in space and time with a gradient of $h_2 = 1000.0$ and using tolerances of $1 \times 10^{-7}$ and $1 \times 10^{-9}$ respectively (run 45).
Figure 21. Linear Stagnation Enthalpy Gradient and Velocity Profile at $H_o = 1000.0$.

The solution converges completely in 9h:18m:28s.
Figure 22. Case2 $u$ Velocity Contour Plot on (61x101) “Standardized Grid”. $h_o = 1000.0$

Case2 is rerun with a stronger gradient. $h_o = 3000.0$ on the same standardized grid.
Figure 23. Linear Stagnation Enthalpy Gradient and Velocity Profile at $h_0 = 3000.0$.

The solution converges completely to a tolerance of $1 \times 10^{-7}$ in space in 0h:9m:50s (run 50). The $u$ contour plot, Figure 24 is significantly different compared with Case1, Figure 20.
Figure 24. Case2 $u$ Velocity Contour Plot on (61x101) "Standardized Grid". $h_o = 3000.0$.

There appears to be a strong region of flow interference at the idim boundary. The flow gradient is so strong that it produces a boundary-layer thickness which approaches the limits of the computational domain. Therefore, it is not appropriate to solve for this gradient strength with the current grid size.

Closer inspection of the Case2 computational domain and the selection of boundary conditions reveals an error at the idim boundary. The previous Case1 and Case2 runs were computed using a fixed at freestream boundary condition at the idim, the top of the domain (idimbc=-1).

However, the definition of the linear gradient for Case2 sets the local tangential velocity as $u \gg U_x$ at this boundary. Therefore this boundary condition is incorrect for Case2 computations. The result of the fixed at freestream boundary condition is clearly evident in Figures 22 and 24 where the idim boundary shows a region of slower moving fluid with respect to the initial flowfield gradient strength.

As an initial boundary condition modification, the top of the computational domain is set to first-order computation from the interior cells (idimbc=-3).

To provide further insight to the flow gradient effects, computation runs are performed on Case3 for the nonlinear stagnation enthalpy gradient. With the first-order interior boundary condition at idim, Case3 was solved on the same (61x101) size grid with $y_{max}=0.001$ meters. The solution is attempted in space and time for values of a gradient defined by $C=-150$, with convergence criteria of $1 \times 10^{-7}/1 \times 10^{-9}$. The space block converges completely, however the time block is divergent.
with CFL=-20 (run 51). Relaxing the gradient strength to $\sigma=-100$ and the convergence criteria to $1\times10^{-7}/1\times10^{-8}$ results in a complete space-time converged run for Case3 (run 56). The $u$ velocity contour plot, shown in Figure 25 indicates strong boundary-layer/grid size interference at the idem boundary. Based on the contour plots shown for Case2 and Case3 in Figures 20, 22, and 23, it is apparent that the allowable gradient strength is restricted by the computational domain, requiring appropriate selection of gradient versus $v_{max}$.

![Case3 u Velocity Contour Plot](image)

Figure 25. Case3 $u$ Velocity Contour Plot on (61x101) “Standardized Grid” $\sigma=-100$.

Close inspection of the flowfield properties for Case3 reveal oscillations in the tangential, $u$ velocity component in the outer flow region. Since the second-order effects of interest for this study are potentially small and difficult to isolate, it is important to determine the cause and significance of these oscillations. Further inspection reveals the same oscillations in Case1 and Case2 computational results. A sample of the $u$ velocity component oscillations in the external flow region is provided in Figure 26. The velocity profiles at six “k” grid stations along the plate length are presented for the outer flow. The velocity oscillates increasingly with “k” station.
In summary, at this point in the study, successful boundary-layer results have been obtained for the Case1 baseline flow. Figures 9 and 15 both show results which indicate a well defined, stable boundary-layer. The Case2 computations depict larger absolute boundary-layers, as expected with the linear velocity gradient at the leading edge. However, the choice of gradient strength is mitigated by the height of the computational domain. Furthermore, the relative comparisons of boundary-layer parameters indicate no significant second-order effects, particularly with respect to the displacement thickness. Case2 computational data has been obtained for variations in gradient strength of \( h_0 = 0.5, 10, 100, 1000, 3000 \). At this point no significant conclusions can be drawn from the limited computational data for Case3.

However, several repetitive ambiguities are evidenced for all of these runs. The inconsistent flow state at the left \( k_u \) boundary shows a serious inconsistency in the computational interpretation of the physical model. The flow oscillations observed in the outer flow region indicate possible solution
instabilities and raise questions concerning the solution accuracy and feasibility of quantifying second-order flow effects.

Since these problems are evident in the solutions for all three Cases they seem most likely to be the result of an inadequate grid design. Closer inspection of the grid design reveals variably "skewed" aspect ratio cells in critical flow regions. Therefore, computational efforts are now focused on developing an improved grid design through aspect ratio modifications.

**Grid Aspect Ratio Modifications / idimbc = -5 tests**

"idim" Boundary Condition

Based on further analysis of the physical flow conditions it is apparent that the selection of the idim boundary condition is incorrect. Previous runs were setup assuming the top grid boundary to be fixed at freestream conditions. However, with gradient flows the idim boundary is not constant at the baseline, Case1 freestream flow conditions. The gradient is designated such that the flow at the plate surface is set at \( u=U_\infty \) with the velocity gradient increasing linearly (for Case2) from the plate surface to the idim grid boundary.

Additionally, the boundary-layer flow is characterized by displacement of the external flow streamlines through some distance by the growth of the boundary-layer. This displacement, measured through the displacement thickness parameter, must satisfy the law of Conservation of Mass. Therefore, the external flow streamlines which are displaced will propagate the flow displacement through the flowfield up to the idim boundary, potentially resulting in a small \( v \) velocity component greater than zero at this boundary. Fixing the flow to freestream, with \( v = 0 \), therefore violates this physical flow response to the boundary-layer. While the choice of idimbc=-3, first-order from interior cells, is acceptable, it is more appropriate to select a subsonic inflow/outflow boundary condition (idimbc=-5). Therefore, for the remaining computations of all flow cases the idim boundary condition is set to subsonic inflow/outflow unless otherwise specified.
Grid Extension Design

The incorrect velocity profile at the \( k_o \) boundary requires attention to isolate its cause and implications on the flow solution. In an attempt to eliminate the problem, a small grid extension is attached in front of the plate, to provide an initial region of fully resolved freestream flow and eliminate the possibility for a flow gradient at the plate leading edge. The extension, grid zone1, is \((61 \times 11)\) grid points over 0.1 meter, using the same compression \( \beta \) of 1.0005 normal to the plate.

Grid compression in the flow direction \((k)\) is added to the main computation grid, grid zone2, at the plate leading edge. The intent of the compression is to improve the resolution at the \( k_o \) boundary and thus eliminate the incorrect velocity gradient evident at \( x=0.0 \).

This grid is executed as a GASP multi-zone run, solving zone1 in space prior to solution of zone2. Zone2 is solved in space and time to a resolution of \( 1 \times 10^{-7} / 1 \times 10^{-5} \). The run converges to the selected tolerances in space but is not successful for the time convergence. Examination of the contour plots for the \( u \) velocity component reveals a small boundary-layer flow developing in zone1 at the \( i_o \) boundary. However, the zone1 grid is setup with no wall boundary conditions and should therefore produce no boundary-layer. When zone1 is solved independently of zone2, in an isolated case, the results are correct, with no "boundary-layer" at the \( i_o \) boundary.

Fine/Coarse/Fine/Coarse Grid Design

The lack of success with the grid extension design requires a complete redesign of the grid to provide highly resolved grid regions where necessary, and coarse regions where possible to reduce computational expense. The new grid is designed such that the grid is composed of four distinct solution regions. The first "fine" mesh region is intended to accurately resolve the initial boundary-layer growth, when flow gradients will be very high. The following region is an area of less flow activity since the boundary-layer should be fully developed by this point. Therefore that grid section is "coarse" to improve computational time. The third section, defined with a fine mesh, is designed to clearly resolve the flowfield in the analysis region, from approximately \( x=0.2 \) (m) to \( x=0.7 \) (m). The final section is designed as a coarse mesh since it is beyond the analysis region and will not
influence the second-order effects. A diagram of the grid design, showing the sections and breakpoints is provided in Figure 27.

Figure 27. F/C/F/C Grid Design Diagram with Break-Points.

Controlling the cell aspect ratio is deemed critical in certain flow regions, particularly at the left boundary of the grid and along the plate surface. Therefore selection of the grid stations which defined each region requires analysis of the resulting effects on cell aspect ratios. For this analysis the cell aspect ratio is defined as:

\[ AR = \frac{\Delta y}{\Delta x} \]  

Equ. (64)

Therefore, large aspect ratio cells are "tall" whereas small aspect cells are "short" and stretched.

Samples of these "poor" aspect ratio cells are given in Figure 28.
Cell Aspect Ratio Analysis

Aspect Ratio Effects
In addition to the modification of the idim boundary condition, close inspection of the current grid characteristics reveals poor aspect ratio grid cells in critical flow regions. The left grid boundary shows small aspect ratios while the external flow region in the analysis section shows large aspect ratios. The physical nature of high-speed, boundary-layer flows requires strong grid stretching to resolve the typically steep velocity gradients near the wall. Such a grid design fosters the small aspect ratio cells near the wall. The addition of stretching in the streamwise, x direction produces large aspect ratio cells at the left boundary and small cells in the external flow regime.
The solution exhibits oscillations in the external flow region, possibly caused by poor aspect ratio cells. The effects of grid aspect ratio on numerical stability and convergence have been addressed most recently by Beulow, Venkateswaran and Merkle.\textsuperscript{40} They investigated the primary cause of aspect ratio convergence problems and developed an approach for consistent, numerical convergence of the N-S equations with any grid aspect ratio. Their efforts involve a systematic stability analysis of a preconditioned form of the governing (Euler and N-S equations). The preconditioned Navier-Stokes equations were previously shown to converge for regular-sized grids for a wide range of Mach and Reynolds numbers.

The results of the stability analysis of Beulow, Venkateswaran and Merkle revealed a problem with the standard local time-step calculation employed in most CFD solvers, including GASP. Typically, the analyst selects a constant CFL condition, thereby allowing for variable time steps based on the local grid size. For most explicit numerical schemes which solve hyperbolic Partial Differential Equations the CFL condition is:

\[
\left| \frac{c \Delta t}{\Delta x} \right| \leq 1 \tag{65}
\]
where $\Delta t$ is the time step, $\Delta x$ is the space step and $c$ is the wave speed. The minimum time step is selected in the two coordinate directions (maximum CFL). However, the stability results show for grid with non-unity Aspect Ratio cells the optimum time-step choice should be based on the maximum, not the minimum. For the Navier-Stokes equations the additional criteria of selecting the maximum von Neumann number (VNN) is required. The authors refer to the combined time-step requirement as the “min-CFL, max-VNN time-step”. For flat plate laminar boundary-layer flow their results showed a convergence rate improvement of approximately 20 times versus the conventional, “max-CFL, max-VNN”, method.

$$\Delta t = \min\left(\frac{CFL \Delta x}{\lambda_x}, \frac{CFL \Delta y}{\lambda_y}\right) \quad \Delta t = \max\left(\frac{CFL \Delta x}{\lambda_x}, \frac{CFL \Delta y}{\lambda_y}\right)$$

Equ. (66)

where the lambda terms are defined as the acoustic eigenvalues in each direction. The authors tested their algorithms on several flow cases, including laminar flat plate boundary-layers at $Ma=0.1$. For one of the laminar flow cases, at $Re=4 \times 10^5$, the convergence rate was improved by a factor of 20 using their algorithm versus the conventional approach.

Therefore it is evident that accurate, timely convergence of the second-order boundary-layer flows is strongly dependent on the grid design, particularly the aspect ratio selection.

**Grid Compression Level Tests**

As discussed earlier, the fine, coarse, fine, coarse (“fcfc”) grid design is designed to address the cell aspect ratio problem. In addition to the distribution of mesh stretching, the grid size is modified to include more grid points, providing for a smaller possible $\Delta x$ and thus improved aspect ratios. The new grid is designed using $61 \times 401$ cell points. Mesh stretching is initially setup to provide equal stretching in the normal and streamwise directions based on the value of $\beta$. This condition ensures very good cell aspect ratios at the plate leading edge, in the critical stagnation point flow region. The initial choice of the grid stations for the mesh sections is shown in Table 6.
Table 6.

Initial Grid Zone Breakpoints

<table>
<thead>
<tr>
<th>k breakpoints</th>
<th>x breakpoints</th>
</tr>
</thead>
<tbody>
<tr>
<td>k1 = 61</td>
<td>x1 = 0.05</td>
</tr>
<tr>
<td>k2 = 81</td>
<td>x2 = 0.3</td>
</tr>
<tr>
<td>k3 = 381</td>
<td>x3 = 0.6</td>
</tr>
<tr>
<td>k4 = kdim</td>
<td>x4 = xmax</td>
</tr>
</tbody>
</table>

Computational runs are performed using the test grid at varying levels of compression to a low convergence tolerance. The intention is to quickly determine the improvements in resolution using variations in the stretching parameter. Three test runs are performed with decreasing values of $\beta$;

- $\beta = 1.0005$, Test 1 (run 59)
- $\beta = 1.00005$, Test 2 (run 61)
- $\beta = 1.000005$, Test 3 (run 62)

As an example of the overall “fcfc” grid design, the grid for Test 1 is shown in Figure 30.
The plots in Figure 31 show the $u$ velocity for the three compression tests for several cells near the plate stagnation point. The results indicate that reducing the stretching parameter, and thus compressing the mesh points closer to the plate surface, results in significantly improved boundary-layer resolution.

Based upon this analysis, it appears that high numerical resolution could be obtained with very fine mesh compression at the plate surface. This is true for the resolution of the boundary-layer.
inner flow region. However, the cell aspect ratio in the external flow region would become very large. The intent of our computational analysis is to isolate the second-order interaction of the boundary-layer with the external flow and large aspect ratio cells are detrimental to the resolution of the interaction effects. As a compromise between improved boundary-layer resolution and acceptable aspect ratios in the external flow region, the computational grid is redesigned with $\beta=1.0000005$ and $idim=101$. The streamwise (x direction) grid stretching is removed to improve the cell aspect ratios in the interface region in section 1. As a result of the previous computational efforts, ymax is increased from 0.001 (m) to 0.05 (m) to reduce the interference of the computational domain height with the boundary-layer development. Figure 32 is the modified “fcfc” grid used in the following computational runs.

![Figure 32. Case2 Velocity Contour Plot on (101x401) “Standardized Grid”, $h_0^* = 1000.0$.](image)

Additionally, for the gradient flow cases, the initial flow profile is modified. For the top 20% of the grid height (ymax), the gradient is fixed at a constant value of $h_0$, resulting in the general gradient shapes shown in Figure 33 for Case2 and Case3.
Computational Runs ("fcfc" Grid)

To provide a baseline solution, a computational run is performed for Case1 on this new grid. The solution converges to a tolerance of $1 \times 10^{-7}$ (run 67). A series of computational runs are then performed for Case2 with the same grid design. Initially, Case2 is run with $\hat{h}_o = 1000.0$ for convergence tolerance=$1 \times 10^{-7}$. 

Figure 33. General Shape of Gradients with Constant $\hat{h}_o$ for Top 20% $\gamma_{\text{max}}$. 
The run is divergent at plane 392 (run 66). The $u$ velocity contour plot for this run, Figure 35, indicates the computational domain is severely interfering with the boundary-layer growth.
With the gradient strength of $h'_{\phi} = 1000.0$, the boundary-layer is not fully developed within the present height of the domain, $y_{\text{max}}=0.05$ (m). The gradient strength is therefore reduced to $h'_{\phi} = 25$ and run with idimbc=-1 and the same tolerance criteria.

![Graph showing stagnation enthalpy and velocity gradient with $h'_{\phi} = 25$.](image)

However, this solution fails to converge plane 214 (run 68).

To isolate the cause of the divergent planes for these Case2 computation runs, the previous Case2 test run is performed with $h'_{\phi} = 25$ but with idimbc=-3, first-order extrapolation from interior cells. The tolerance criteria is again set to $1 \times 10^{-7}$. The run completely converges in 0h:14m:02s.
(run 69). However, examination of the $u$ velocity contour plot, Figure 37, reveals a very constant boundary-layer with little development.

Therefore it appears that the selection of idimbc=-3 provides the stability to converge the solution to the $1 \times 10^{-7}$ tolerance. However, idimbc=-3 results in a coarsening of the boundary-layer resolution. Apparently, the vorticity which results from the Case2 velocity gradient is not properly modeled for the idim boundary with the idimbc=-3 condition. As discussed previously, the subsonic inflow/outflow condition is a more accurate representation of the physical flow conditions at this boundary.

Therefore, the remaining computational runs are performed with subsonic inflow/outflow at the idim boundary (idimbc=-5).

Relaxing the tolerance criteria from $1 \times 10^{-7}$ to $1 \times 10^{-6}$ for the computation run with $h_\alpha = 25$ and idimbc=-5 results in a divergent solution at plane 48. Therefore the tolerance criteria is increased to $1 \times 10^{-5}$ with the intention of improving the flow resolution prior to the divergent plane, thus controlling the solution stability. The solution is again divergent on plane 213.

To improve the convergence for these Case2 solution runs, the computational region is increased in height from $y_{\text{max}}=0.05$ (m) to $y_{\text{max}}=0.1$ (m). The intention is to reduce the interference of the gradient flow with the idim boundary. With $h_\alpha = 100.0$ the gradients are altered as shown in Figure 38. The solution is attempted with a convergence tolerance of $1 \times 10^{-7}$ and idimbc=-5, resulting in a divergence at plane 393 (run 74).
The results to this point seem to indicate that the vorticity effect is causing a strong solution instability, resulting in divergent Case2 solutions. To minimize the influence of the computational boundaries on the solution stability, the gradient is redefined to ensure a larger region of constant flow in the vicinity of the idim boundary. This modification and the grid stretching in the i direction restrict the nonuniformity to a smaller percentage of the external flow region. The original definition of the linear stagnation enthalpy gradient fixes the top 20% of the computational domain to a constant local tangential velocity, \( \dot{u} \), as shown in Figure 38. The restriction for the velocity gradient is modified to fix the top 20% of the domain with respect to idim instead of \( y_{\max} \). A run is performed with \( \dot{h}_o = 25 \), \( y_{\max} = 0.05 \text{ m} \) and the convergence tolerance of \( 1 \times 10^{-7} \). The solution
diverges at plane 396, with results consistent with the previous runs. However, the gradient is maintained as fixed constant for 20% of idim for the remaining Case2 runs.

The convergence of Case2 on the “fcfc” grid is unsuccessful up to this point. The variations of gradient strength and tolerance settings do not improve the convergence problems. Therefore the grid design is further investigated for clues to the instabilities encountered for Case2.

**Uniform Grid Design**

The current grid still contains poor aspect ratio cells in the first, fine mesh section. Since the nonuniform/uniform grid presents a serious challenge for maintaining proper cell shapes it is necessary to consider a completely uniform grid design. The uniform design provides a simpler approach to maintaining acceptable aspect ratio cells, since the cell shape is controlled through the selection of mesh section break-points. To provide the uniform grid design the stretching is removed in both the normal and streamwise flow directions. The overall grid size is increased to (101x701) to provide improved aspect ratios while maintaining a large number of cells in the normal direction for adequate boundary-layer resolution. The grid dimensions are set to $x_{\text{max}}=1.0 \, \text{(m)}$, $y_{\text{max}}=0.05 \, \text{(m)}$.

The aspect ratio (AR) for the uniform grid can be defined as

$$AR = \frac{y/L_y}{x/L_x} \quad \text{Equ. (67)}$$

where

$L_x$, $L_y$=number of grid lines in $x$, $y$-direction respectively
$x=$max length of grid section
$y=$max. height of grid ($y_{\text{max}}$)

Given a fixed grid height, $(y=y_{\text{max}}=0.05 \, \text{(m)})$ and a fixed number of normal grid lines for resolving the boundary-layer $(L_y)$ the aspect ratio of cells within a given section is driven by the number of streamwise grid lines in the section $(L_x)$, and the section length, $(x)$. For the increased grid size, $(101x701)$, the section break-points are redefined as shown in Table 7.
Table 7.

Redefined Grid Zone Stations

<table>
<thead>
<tr>
<th>k break-point</th>
<th>x break-point</th>
<th>Aspect Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>kone=101</td>
<td>x one=0.05</td>
<td>AR\textsubscript{1}=1.0</td>
</tr>
<tr>
<td>ktwo=181</td>
<td>x two=0.3</td>
<td>AR\textsubscript{2}=0.16</td>
</tr>
<tr>
<td>kthree=681</td>
<td>x three=0.6</td>
<td>AR\textsubscript{3}=0.833</td>
</tr>
<tr>
<td>kfour=701=kdim</td>
<td>x four=1.0=x\text{max}</td>
<td>AR\textsubscript{4}=0.025</td>
</tr>
</tbody>
</table>

The break-points are selected to optimize the cell aspect ratio in sections one and three, (AR\textsubscript{1}, AR\textsubscript{3}), the critical flow regions.

A test run is performed on this grid for Case2, with \( h^* = 1000.0 \) (run 76).
Figure 39. Enthalpy and Velocity Gradient with $h_0' = 1000.0$ on Uniform Grid.

The run is solved in space to a convergence tolerance of $1 \times 10^{-7}$, and is halted at plane 124 to investigate the resolution of the boundary-layer in section one, near the stagnation point. Figure 40 shows the $u$ contours plot overlaid with flow streamlines. The boundary-layer appears to be large and under-developed within the height of the computation domain. A strong $v$ component and the associated streamline swirling are evident near the idim boundary.
For comparison with the Case2 results, a Case1 solution is obtained for the same number of planes (run 77). The resulting boundary-layer is smaller and restrained well within the computational domain limits. Also note the absence of a strong normal velocity component at the idim boundary, indicated by the steady, parallel flow streamlines in Figure 41.
This solution comparison indicates that the strong enthalpy gradient caused a freestream vorticity resulting in a large $\nu$ velocity component at the idim boundary. The gradient strength may therefore still be interfering with the flow, preventing the resolution of significant second-order effects. A Case 2 computation is therefore run with a weaker gradient, $h'_{\phi} = 50.0$, on the same grid (run 78).

The resulting gradients are shown in Figure 42. This solution is divergent at plane 163.

![Figure 42. Enthalpy and Velocity Gradient with $h'_{\phi} = 50.0$ on Uniform Grid.](image)

Since the solutions are still divergent for Case 2, further grid modifications are made. The grid breakpoints are redefined to improve the aspect ratios in the coarse sections, AR$_2$ and AR$_4$. 
Table 8

Modified Grid Zone Stations

<table>
<thead>
<tr>
<th>k break-point</th>
<th>x break-point</th>
<th>Aspect Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>konc=101</td>
<td>x one=0.05</td>
<td>AR₁=1.0</td>
</tr>
<tr>
<td>ktwo=281</td>
<td>x two=0.2</td>
<td>AR₂=0.59</td>
</tr>
<tr>
<td>kthree=981</td>
<td>x three=0.6</td>
<td>AR₃=0.866</td>
</tr>
<tr>
<td>kfour=1001=kdim</td>
<td>x four=1.0=xmax</td>
<td>AR₄=0.025</td>
</tr>
</tbody>
</table>

The grid size is increased to (101x1001) and the sections are redefined as shown in Table 8. Again, Case2 is run with the weaker gradient, \( h_0 = 50.0 \) on this modified grid. The solution diverges at plane 802. The \( u \) contour plot depicts boundary-layer interference with the domain height starting at \( x=0.2 \) (m) (run 79).

![Figure 43. \( u \) Velocity Contour Plot (Case2, \( h_0 = 50.0 \)).](image)

The gradient strength is reduced even further to \( h_0 = 5.0 \), giving the gradients in Figure 44.
The solution is not divergent, however it is unstable (run 80). Plane 525 does not converge to the required tolerance, and the residual stops decreasing. Examination of the data, in Figure 45, seems to indicate the possible cause for convergence problems thus far could be the interference of the vorticity effect with the $h^+$ boundary.
At this point an error with the grid design is evident. Close examination of the plots for the Case2 runs shown in Figures 43 and 45, reveals the top of the computational domain is not properly set at \( y_{\text{max}} = 0.05 \) (m). An error in the grid generation routine causes \( y_{\text{max}} = 0.0495 \) (m). Modifications to the grid generation routine fix the mesh and the run is tested again, resulting in the same divergent solution as shown in Figure 45.

The computational results using this uniform grid design indicate that while the cell aspect ratio is near unity for all critical flow regions, the number of computational points in the boundary-layer, inner flow region is low. Based on the uniform, clean flow conditions of Case1, the expected boundary-layer thickness is predicted to be within 0.001 (m). Figure 46 is a blowup of the uniform computational grid near the plate leading edge.
As shown in Figure 46, the grid is designed to provide only two grid cells within the boundary-layer. Therefore, it is unlikely that the boundary-layer or the interaction effects are accurately resolved.

"Visualization Layer"

The GASP post-processing routine, "PRINT" provides the capability to interpolate the flow data to cell nodes or to cell centers. The previous data plots are interpolated to nodes. However, for the results of the previous Case2, $h' = 50$, computation (run 82) given in Figure 43, TECPLOT is used to compare the visualization of data interpolated to nodes and cell centers. A comparison of two such plots is shown in Figure 47. The significance of the interpolation method is observed by comparing the boundary-layer region of both plots.
The interpolation to nodes results in an even, level layer of flow with little growth downstream. The interpolation to cell centers provides the expected smooth boundary-layer growth. The TECPLOT interpretation of the cell node data results in a "visualization" layer, due to the large velocity gradient between the plate surface (no-slip) and the first grid line. However, with cell centered data the velocity components are output as their "true" values, given the finite-volume approach used by GASP. Therefore TECPLOT properly depicts the gradients at the plate surface. The result is a k_{0} boundary which does not depict the arbitrary velocity gradient evidenced for the cell-node data.

**Uniform Grid With Vertical Sections (Velocity Oscillations)**

In an effort to maintain good aspect ratios and a large number of grid points in the inner region, the uniform grid is modified to include two normal mesh sections in addition to the four streamwise mesh section. The intent is to provide a normal break-point, similar to the method used in the streamwise direction. The first vertical section provides a region of compressed mesh near the plate surface to resolve the flow gradients in the boundary-layer region. The second vertical section serves as a coarser flow region for quickly resolving the simpler, external flow region. The inner region is purposefully designed to extend beyond the expected boundary-layer height to provide...
improved resolution at the boundary-layer/external flow interface, thus capturing the second-order interaction effects.

Once again, the grid size is increased to (101x1201) and the break-points are redefined as in Table 9.

Table 9.

Modified Grid Zone Stations (BLHGT=0.003 (m), BLPTS=11)

<table>
<thead>
<tr>
<th>k breakpoint</th>
<th>x breakpoint</th>
<th>Aspect Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>kone=83</td>
<td>x one=0.05</td>
<td>AR_1=0.5</td>
</tr>
<tr>
<td>ktwo=233</td>
<td>x two=0.2</td>
<td>AR_2=0.6</td>
</tr>
<tr>
<td>kthree=899</td>
<td>x three=0.6</td>
<td>AR_3=0.5</td>
</tr>
<tr>
<td>kfour=1201=kdim</td>
<td>x four=1.0= xmax</td>
<td>AR_4=0.37</td>
</tr>
</tbody>
</table>

The vertical sections are defined by two new break-points, BLHGT/BLPTS. These parameters specify the “expected” boundary-layer height and the desired number of boundary-layer points. Therefore, the aspect ratios, as shown in Table 9, are calculated with y^=BLHGT=0.003 (m).

An initial computational run on this grid for Case2 with $\dot{h}_0 = 5.0$ diverges at plane 475 (run 89).
Figure 48. Enthalpy and Velocity Gradient with \( h_\circ = 5.0 \) - BLHGT=0.003 (m)/BLPTS=11.

Examination of the resulting flowfield shows oscillations in the \( u \) velocity component in the external flow region. The resulting boundary-layer height is under \( y=0.001 \) (m), allowing for a decrease in BLHGT.

For the next test run, the vertical section is reset to BLHGT=0.0015 (m). The computation is performed with \( h_\circ = 0.5 \).
Figure 49. Enthalpy and Velocity Gradient with $h_0 = 0.5$ - BLHGT=0.0015 (m)/BLPTS=11.

The solution is halted at plane 160 (run 90). The output again shows flow oscillations in the external flow. In an attempt to dampen the flow oscillations the gradient is decreased to $h_0 = 0.2$
Figure 50. Enthalpy and Velocity Gradient with \( h_0 = 0.2 \) - BLHGT=0.0015 (m)/BLPTS=11.

The computation is attempted with an increased tolerance of \( 1 \times 10^8 \). The solution diverges at plane 1033 (run 93).
A baseline run is performed with Case 1 (run 91) on this new vertical sectioned uniform grid. The case converges completely in space to a tolerance of $1 \times 10^{-7}$. Comparisons with the partially converged solution obtained for Case 2 (run 93) indicate the normal velocity component, $v$, is much weaker for the baseline, clean flow.
Close study of the flow velocity oscillations indicate the effect is evident for both Case1 and Case2 solutions. Therefore, the oscillations are obviously a problem with the flowfield resolution. However, given the small magnitude of the velocity oscillations, generally of order 9(-3), we can assume they will not adversely affect the development of second-order flow effects. Since these oscillations occur well within the external flow region, far from the interaction region, we can proceed to resolving the flow without focusing on eliminating the problem. Furthermore, since the oscillations are negligible, it is not critical to maintain the cell aspect ratio in the outer region. Instead the grid design will focus on obtaining excellent resolution in the inner flow and interaction regions.

It is difficult to maintain good aspect ratio cells throughout the computational domain and simultaneously restrict the grid size to machine limitations. Therefore, while a uniform grid design is desirable for easily specifying cell aspect ratios, it does not provide a feasible approach for compressing the grid in computationally critical regions (e.g. near the plate surface).

Nonuniform Grid Design (“Final” Grid)

As discussed above, the flow velocity oscillations are minimal. Therefore, the grid is redesigned with the intention of attaining high flow resolution in the boundary-layer and interface regions, without concern for the cell aspect ratio in the external flow region.

The final grid design provides a highly compressed region of flow well within the boundary-layer. The results analysis region is reset to 0.2 ≤ x ≤ 0.8, to reduce the streamwise grid coarseness in the trailing edge section. Since the cell aspect ratios are not critical, the slight reduction in streamwise compression of the analysis region is acceptable. The grid design is based on the expected value of the boundary-layer thickness at x=0.6 meters. With the expansion of the analysis region, x=0.6 (m) is within the region of interest and the grid is structured to the boundary-layer height at that location. Using Equation 52,

\[
\frac{\delta}{x} = 5Re_x^{1/2}
\]
the expected boundary-layer thickness at \( x=0.6 \) (m) is calculated as \( 3.682 \times 10^{-4} \) (m).

Based on the approximation for \( \delta_{x=0.6} \), the overall grid height is reset to \( y_{\text{max}}=0.004 \), approximately \( 10 \delta_{x=0.6} \), and the grid size is maintained at (101x701). The grid is designed to fix 40 points within the \( x=0.6 \) boundary-layer height, using exponential grid stretching from the plate surface. Beyond the boundary-layer edge, the grid is reset to uniform spacing, with 35 points over a distance of another \( 2 \delta_{x=0.6} \). The remaining 25 points are distributed over the remaining normal distance. The gradient is designed to fit the nonuniform portion within a distance of \( 5 \delta_{x=0.6} \) normal to the plate. The intention is to provide a strong region of nonuniformity well within the overall computational domain height. A graphical representation of this "final" grid design is shown in Figure 53. While the grid design is based on \( \delta_{x=0.6} \), the grid height \( (y_{\text{max}}=0.004) \) is \( 6.5 \delta_{x=1.0} \), allowing for adequate boundary-layer development at the end of the plate.

Figure 53. Final Grid Design for All Cases.

The "fcfc" streamwise grid zones are defined with the breakpoints given in Table 10.
Table 10.

Modified Grid Zone Stations for “Final” Grid

<table>
<thead>
<tr>
<th>k breakpoint</th>
<th>x breakpoint</th>
</tr>
</thead>
<tbody>
<tr>
<td>kone=75</td>
<td>x one=0.05</td>
</tr>
<tr>
<td>ktwo=150</td>
<td>x two=0.2</td>
</tr>
<tr>
<td>kthree=600</td>
<td>x three=0.8</td>
</tr>
<tr>
<td>kfour=701=kdim</td>
<td>x four=1.0= xmax</td>
</tr>
</tbody>
</table>

Computational results for the “final” grid design are presented in the following chapter, Flowfield Cases - Results.
FLOWFIELD CASES - RESULTS

Inviscid Validation Solution

Computation Description
Validation of GASP with a simple, inviscid flowfield provides assurance of the computational stability of the CFD solver. Using the flat plate physical geometry, the computation is performed assuming entirely inviscid flow. Assuming no viscous flux contributions in any of the flow directions ensures inviscid conditions and instructs GASP to use the inviscid Euler equations for the computational solution. This requires the values of the \text{visflx}_i, \text{visflx}_j, \text{and} \text{visflx}_k \text{input variables equal zero in the gasp zone file. The resulting numerical solution should be completely uniform flow with no boundary-layer development.}

For this computation run the “final” grid is used with (701x101) cells and \(\gamma_{\text{max}}=0.004 \text{ (m)}.\)

The solution converges to a tolerance of \(1x10^{-7}\) in 0h:05m:34s (run 123).

Inviscid Results
The \(u\) velocity contour plot produced by the GASP inviscid solution is provided in Figure 54.

The flowfield is completely uniform at freestream conditions.
Therefore GASP yields the expected inviscid flow solution for the validation run. The results ensure that GASP is producing consistent computational solutions for simple flows, providing the foundation for the complex viscous computations of this study.

**Case 1 (Baseline Uniform)**

**Computation Description/Results**

CFD solutions of a uniform flowfield over a flat plate provide the baseline boundary-layer upon which to make later judgments regarding the influence of second-order effects. The overall results of the final Case 1 solution show a simple, steady boundary-layer development.

The first Case 1 computational run includes an error in the grid generation routine, which sets $y_{\text{max}}=0.002$ (m) instead of 0.004 (m) as is intended. However, the solution converges in space to a tolerance of $1\times10^{-8}$ (run 97). A follow-up run is performed with the proper grid dimensions of $y_{\text{max}}=0.004$ (m). The solution is convergent in space to a tolerance of $1\times10^{-7}$ (run 98). Attempts to converge the solution on the larger domain to $1\times10^{-8}$ tolerance result in convergence oscillations at plane 11. The results of both runs are examined and compared. The contour plot of the $u$ velocity...
component is shown for the small, $y_{max}=0.002$ (m) grid in Figure 55 and for the large, $y_{max}=0.004$ (m) grid in Figure 56.

Figure 55. $u$ Velocity Contour Plot for Case1 on Small Domain “Final” Grid.

Figure 56. $u$ Velocity Contour Plot for Case1 on Large Domain “Final” Grid.
The boundary-layer resolution on the smaller domain is obviously much stronger and more defined than the larger grid. This improved boundary-layer resolution is primarily attributed to the improved convergence tolerance for the smaller domain. Both grids are designed to fix 75 points within the boundary-layer region, as depicted in Figure 53, regardless of \( y_{\text{max}} \). Therefore, the primary difference in mesh spacing between these two grids occurs beyond \( y = 3 \delta_e \), well into the external flow region. The external flow region is therefore much more tightly meshed for the smaller grid. The tighter mesh improves the computational stability in the external flow region, reducing oscillations and thus increasing the convergence tolerance and resulting flow resolution.

As a result of the computational improvements with the smaller grid, the "final" grid is modified by setting \( y_{\text{max}} = 0.002 \) (m). This modification is depicted in Figure 57. While the modified grid design is based on \( \delta_e = 6 \), the selection of \( y_{\text{max}} = 0.002 \) corresponds to \( 3.3 \delta_e = 1.0 \) and provides sufficient grid height to adequately resolve the boundary-layer at the plate trailing edge.

![Diagram](image)

**Figure 57** Modified Final Grid Design for All Cases (\( y_{\text{max}} = 0.002 \) (m))

For analysis and comparison with the other flow cases, the baseline Case 1 solution is rerun with the grid design in Figure 57. The solution again converges to \( 1 \times 10^{-8} \) tolerance (run 127).
The baseline flow solution is further quantified by examining the skin-friction coefficient and heat transfer along the plate surface. The skin-friction coefficient within the analysis region for the small grid solution, is shown in Figure 58. The results are plotted against the first-order exact solution for the flat plate geometry given by Equation (68).

\[ c_f = 0.4696 \left( \frac{2}{Re_n} \right)^{1/2} \]  

Equ. (68)

The skin-friction response is as expected for the boundary-layer flow. There is a large jump in \( c_f \) at the leading edge stagnation point (not shown in Figure 58) and as \( x \) increases the skin friction asymptotically approaches zero.

The results for Case1 closely match the first-order solution, with an overall increase in skin-friction throughout the analysis region. The Case1 solution includes the second-order effects of displacement, not accounted for by the first-order exact solution. Therefore, the increase in skin-friction for Case1 may be partially attributed to displacement effects.

The Stanton number, a nondimensional expression for the heat transfer coefficient, is given for the Case1 results in Figure 59 as a baseline for comparison with the enthalpy gradient solutions. The first-order exact Stanton number for the flat plate is given by Equation (69).
The Case 1 Stanton number closely follows the results of the first-order solution. There is a measurable heating of the flow due to the shear forces along the plate surface. The heating exponentially decreases downstream of the plate leading edge.

Figure 59. Heat Transfer Coefficient (St) Along Plate Surface (Case 1).

The small increase in Stanton number for Case 1 can again be attributed to second-order displacement effects not accounted for by the first-order exact solution.

Comparison with Theory (Blasius Solution)

Direct comparison of the Case 1 results with the theoretical results provided by the Blasius solution indicate good correlation. Figure 60 depicts the normalized velocity profile within the boundary-layer for the $y_{max}=0.002$ (m) solution at $x=0.575$ meters. The computational solution shows strong agreement with theory when plotted against the Blasius boundary-layer velocity profile using the same normalization.
Case 2 (Linear Stagnation Enthalpy Gradient, \( h_0 = 50.0 \))

Linear Gradient Description

The Case2 computational run on the “final” grid, with \( y_{\text{max}} = 0.002 \) (m), is performed with a gradient strength of \( h_0 = 50.0 \) (run 125). With the grid stretching and dimensions from Figure 57, the gradients are as shown in Figure 61.
Figure 61. Enthalpy and Velocity Gradient with $h_0 = 50.0$ on Modified "Final" Grid.

The solution converges to a $1 \times 10^{-7}$ convergence tolerance with space marching in 0h:13m:43s. The $u$ contour plot, Figure 62 shows a fully-developed, stable boundary-layer.
Comparison with Baseline Uniform Flow (Case 1)

Boundary-Layer Parameters

Comparison of the boundary-layer parameters for the $h_o = 50.0$ Case 2 solution with those for the baseline Case 1 results will reveal potential second-order effects. The boundary-layer, displacement and momentum thickness are depicted in Figure 63. The displacement and momentum thickness are calculated using the compressible form given in Equations (48) and (49).
Figure 63. Comparison of Boundary-Layer Parameters in Analysis Region (Case1/Case2).
The boundary-layer thickness is apparently decreased with the introduction of the flow gradient. The displacement and momentum thickness also both indicate a decrease for the Case2 solution.

**Skin Friction Coefficient/Stanton Number**

Direct comparison of the skin friction and heat transfer coefficients can further reveal the influence of second-order flow effects. As discussed by Emanuel, the theory and historical analysis have shown that:

...if external vorticity (which is essentially "coexistent" with gradients) causes $U_1$ to increase with $n$, then vorticity interaction increases both the skin friction and the heat transfer. In our analysis this would correspond to $h_o$ being positive... As a general conclusion it is relatively self-evident, since vorticity, in this circumstance, is accelerating the flow in the boundary-layer.\(^\text{42}\)

Therefore it is advantageous to compare these parameters for the gradient flow cases against the same parameters for the baseline flow. As is expected, the skin-friction coefficient for the Case2 solution increases in the analysis region, as shown in Figure 64. The gradient is selected so $h_o > 0$, resulting in an increase in $U_1$ with increasing $y$.

![Figure 64. Skin Friction Coefficient Along Plate Surface (Case1 vs. Case2).](image-url)
Examination of the second-order effects on the Stanton number also reveals the expected response.

The heat transfer for the Case2 flow increases with respect to the baseline solution within the analysis region.

![Stanton Number Along Plate Surface (Case1 vs. Case2)](image)

Figure 65. Stanton Number Along Plate Surface (Case1 vs. Case2).

The shape factor for Case1 is determined to be $H=3.86$, while for Case2 $H=3.52$. For laminar flows a shape factor of approximately $H=3.5$ implies an adverse pressure gradient and imminent flow separation. This would imply that both the baseline and Case2 flows are near separation. However, in neither case does the flow exhibit signs of separation. The large Reynolds number for the current freestream conditions, $Re=6.63878 \times 10^7$, may be resulting in adverse flow conditions downstream of the analysis region which are causing separation effects. For this reason it is desirable to analyze a reduced Reynolds number flow later on.

**Case 3 (Nonlinear Stagnation Enthalpy Gradient, $\sigma=-0.029$)**

**Nonlinear Gradient Description**

The Case3 "final" grid run is solved using the nonlinear stagnation enthalpy gradient and resulting velocity gradient produced with $\sigma=-0.029$ and shown in Figure 66 (run 129).
The solution is solved in space to a convergence tolerance of $1 \times 10^{-7}$ in 0h:17m:00s.

**Comparison with Baseline Uniform Flow (Case1)**

**Boundary-Layer Parameters**

Further insight is obtained through comparison and analysis of the Case3 boundary-layer parameters with the baseline Case1 results. Figure 67 shows the comparison. The displacement and momentum thickness are calculated using the compressible form given in Equations (48) and (49).
Figure 67. Comparison of Case3/Case1 Boundary-Layer Parameters
From Figure 67 it is evident that the boundary-layer thickness, displacement thickness and momentum thickness all decrease, relative to the baseline solution, with the introduction of the stagnation enthalpy gradient.

For Case3 the shape factor again implies adverse pressure gradient with separated flow conditions, $H \approx 3.52$. As discussed above for Case2, the separation may occur outside the analysis region, since the Case3 flow does not currently exhibit separation symptoms.

**Skin-Friction Coefficient/Stanton Number**

The skin friction coefficient exhibits the same response shown in Figure 64 for the Case2 results. Within the flow study region, the skin friction increases relative to the baseline solution.

![Graph showing skin friction coefficient along plate surface](image)

**Figure 68. Skin Friction Coefficient Along Plate Surface (Case1 vs. Case3).**

The Stanton number shows the expected increase for the Case3 results versus the Case1 baseline solution.
Reduced Reynolds Number Computations

Computation Description

The previous computational solutions were obtained at a large Reynolds number of $6.63878 \times 10^8$. The shape factor in all cases was approximately $H = 3.5$, indicating potentially separated laminar flow conditions. As a result, new computational solutions are sought at a reduced Reynolds number. The intent is to "stabilize" the flow conditions to improve the shape factor and potentially reduce the signal-to-noise ratio in the solution.

Solutions are obtained for baseline, Case1 conditions and for the Case2 linear stagnation enthalpy gradient with $h_0 = 50.0$. The Reynolds number is reduced by a factor of 50 by limiting the plate length to 0.2 meters versus 1.0 meter and decreasing the density from 1177 kg/m$^3$ to 0.1177 kg/m$^3$. This new physical flow environment gives a Reynolds number of $1.32774 \times 10^5$.

Computational solutions for lower Reynolds number flows with a decreased density of 0.01177 kg/m$^3$ are divergent. Possibly with this low density the GASP solver produces significant numerical error.

To accommodate the smaller plate length the final grid is reduced in size to (101x141). The analysis region is stretched to include the entire plate length, $0.0 \leq x \leq 0.2$. Accordingly, the grid height is based on the expected boundary-layer thickness at $x=0.2$ (m), which gives $\delta = 8.67 \times 10^{-4}$ (m).
Therefore, using the “final” grid design shown in Figure 53, the overall grid height is set to approximately $10 \delta_{x=0.2}$ which gives $y_{\text{max}} = 0.009 \text{ (m)}$. The resulting computational grid is shown in Figure 70.

![Reduced Reynolds Number Computational Grid](image)

**Figure 70. Reduced Reynolds Number Computational Grid.**

For the reduced Reynolds number computations we will examine the pressure, density and temperature profiles at several stations along the plate surface.

**Case 1 Results**

The Case 1 baseline flow converges in space to a tolerance of $1 \times 10^{-7}$ (run 121). The computations were performed with space analysis because time marching produced divergent solutions.
Pressure Profiles

In Figure 71 the pressure profile for Case1 is plotted at several k stations along the plate.

The baseline, freestream pressure is selected as $P_e = 1.01304 \times 10^5$ Pa, as given in Table 2. Therefore, in Figure 71 the effect of the oblique shock wave at the plate leading edge is clearly evidenced by the large jump in pressure around station $k=29$. Further downstream the pressure stabilizes back to freestream conditions.

![Figure 71. Reduced Re# Case1 Pressure Profiles](image)

Density Profiles

Case1 density profiles are shown in Figure 72 within the boundary-layer height for the same k stations. The baseline density for the reduced Reynolds number computations was reduced to $\rho_e = 0.1177$ kg/m$^3$. The density profiles indicate an approximately 74% reduction in the baseline density within the boundary-layer.
Temperature Profiles

The Case 1 temperature profiles are plotted over the entire grid height in Figure 73. The baseline temperature is $T_x=300$ K, as given in Table 2. Examination of Figure 73 indicates the temperature increases approximately 136% within the boundary-layer, to a value of 408K. Outside the boundary-layer the flow quickly cools to freestream conditions. Therefore we see clearly the frictional heating of the flow as a result of the viscous shear forces induced by the plate surface.
Case 2 Comparison with Baseline Uniform Flow (Case 1)

The Case 2 linear stagnation enthalpy gradient flow with $h_{\theta} = 50.0$ converges in space to a tolerance of $1 \times 10^{-7}$ (run 128). The Case 2 solution is directly compared with the Case 1 results presented in Figures 71 to 73.

Pressure Profiles

The initial pressure profile for Case 2 is the same as Case 1, as found from the substitution principle in Equation (27). However, in Figure 74 at station $k=29$, there is a significant increase in the pressure for Case 2 versus Case 1. The increase in pressure observed for Case 1 as a result of the oblique shock at the leading edge is enhanced for Case 2. This is evidence of a stronger shock for Case 2. Further downstream, the pressure decreases quickly.
Density Profiles

The initial Case2 density profile is decreased relative to Case1 due to the choice for the transformation parameter in the substitution principle. Equation (27) and the selection of a positive gradient strength. Figure 75 shows the Case2 density profiles through the entire grid height, indicating a steady decrease in the density downstream of the leading edge. While the Case1 density profiles indicated a decrease in density within the boundary-layer followed by a return to freestream density in the external region, the Case2 profiles do not recover from the boundary-layer decrease. This is primarily the influence of the initial Case2 density profile and thus a result of the linear gradient.
Temperature Profiles

The temperature profiles in Figure 76 show an initially strong jump in flow temperature near the plate leading edge for Case2 relative to the Case1 profile. This temperature steadily decreases to the Case1 temperature profile downstream.
Figure 76. Comparison of Reduced Re\# Case 1 and Case 2 Temperature Profiles.
CONCLUSIONS

Computational experimentation indicates that grid design is a critical factor in the convergence and resulting accuracy of higher-order flat plate boundary-layer solutions. Grid design should focus on providing a refined mesh within the boundary-layer to clearly resolve the steep flow gradients in that region. Concurrently, the grid must be designed to limit flow oscillations in the external region. A grid height of approximately $3\delta$ provides a good balance between these design factors.

The numerical solutions provide accurate resolution of the baseline, supersonic boundary-layer flow over a flat plate. Comparison of the baseline uniform flow CFD solution (Case 1) with the first-order exact solution as observed by Blasius indicates good correlation within the computational resolution of the data. Within the grid analysis region, the Case 1 boundary-layer velocity profile is closely matched to the Blasius profile.

Investigation of the Case 1 skin friction coefficient along the plate surface indicates the expected physical response; an initial large jump in $c_f$ at the stagnation point followed by an exponential decrease downstream. Within the grid analysis region the skin-friction compares well with the first-order exact solution, exhibiting a consistent increase in skin-friction along the length of the plate. Since the first-order solution does not account for the second-order displacement effect, which is captured in the Case 1 numerical solution, this rise in skin-friction is likely a result of the boundary-layer displacement.

The baseline, Case 1 Stanton number also responds as expected along the plate surface. An initially strong heating effect which gradually decreases downstream is the result of the frictional heating of the boundary-layer flow due to the viscous shear forces on the plate surface. Again, the
increase in Stanton number for Case1 relative to the first-order exact solution can be attributed to the displacement effect.

The linear stagnation enthalpy gradient solution (Case2) produces marked increases in both skin-friction and Stanton number within the flow analysis region, 0.2 \leq x \leq 0.8 meters. The observations of Hayes and Probstein support these findings. Additionally, the boundary layer, displacement and momentum thickness all decrease for Case2 relative to the baseline, Case1 solution. However, it is important to consider that the Case2 solution is resolved only to $1 \times 10^{-7}$ tolerance while Case1 results are converged to $1 \times 10^{-8}$ on the same grid. Attempts to converge the Case2 solution beyond $1 \times 10^{-8}$ result in divergence. It is currently unknown if the lower computational convergence of Case2 is of concern. However, the analysis performed here indicates results for Case2 which are supported by theory.

The nonlinear stagnation enthalpy gradient solution (Case3) also shows an increase in skin-friction and Stanton number over the baseline solution. For Case3 the increase in both parameters levels off at approximately $x = 0.4$ (m) while for Case2 the increase is higher and does not level off. This difference demonstrates the variation in second-order effect as a function of the gradient strength and shape. As with Case2, the Case3 solution is converged to $1 \times 10^{-7}$ versus $1 \times 10^{-8}$ for Case1. The significance of the difference in convergence tolerance is currently unknown.

The reduced Reynolds number Case1 and Case2 solutions provide insight to the overall flow characteristics. The gradient appears to strengthen the leading edge oblique shock, as evidenced by the large jump in pressure relative to the baseline solution. The Case1 density is drastically reduced through the boundary-layer but returns to freestream flow conditions in the external region. The Case2 density experiences a similar decrease within the boundary-layer but does not recover, indicating the influence of the density gradient at the leading-edge being transferred downstream. The temperature profiles reveal the effects of the increased heating in the Case2 solution. As observed for the larger Reynolds number run, the linear gradient should produce an increase in Stanton number relative to the baseline flow. The strong jump in temperature for Case2 provides a clear measurement of the heating effect of the gradient. The current analysis did not reveal the effects
of flow separation for the reduced Reynolds number solution. However, the shape factor and boundary-layer parameters were not studied for the lower Reynolds number case and therefore solid conclusions cannot be made as to the flow stability of this solution.

Overall, it is observed that numerical solutions of supersonic, flat plate boundary-layer flows obtained with CFD provide a flexible experimental resource for capturing second-order flow effects. The isolation of individual second-order effects and their particular influence on the flow is difficult since second-order effects are not unique and typically occur simultaneously. However, as demonstrated here, the solution of a simple baseline flow can provide a basis for isolation of the displacement effect and thus allow for measurement of the influence of additional effects in more complex flows. Therefore CFD is potentially an exciting tool for the study of second-order flow effects and for later comparison and qualification of the second-order boundary-layer equations.
RECOMMENDATIONS

Several recommendations can be made for further research beyond the work presented here. Attempts should be made to analyze baseline and gradient flow solutions at the same convergence tolerance. This may require development of a new computational grid to provide stable, convergent numerical results for the gradient flows.

All computational analysis to this point has involved either the Parabolized Navier-Stokes equations (for space marched solutions) or the Thin-Layer Navier-Stokes equations with viscous flux contributions in the normal flow direction, where flow gradients are steepest (for time marched solutions). This simplification was taken since resolution of viscous terms in any direction requires a fine grid mesh in that direction. Computation with the full Navier-Stokes equations would require a fine grid mesh in all directions at all points of the computational domain. This design can result in a grid too large for current hardware capabilities. However, since second-order effects are often small, good resolution of their influence may require the use of viscous contributions from all coordinate directions. Therefore, it is worthwhile to attempt computational solutions of baseline and gradient flows using the full Navier-Stokes equations, if possible.

The influence of the reduced Reynolds number on the resolution of second-order effects was not thoroughly studied here. The behavior observed for the linear gradient flow relative to the baseline flow demands further inspection and analysis. The effects of Reynolds number on the skin-friction and Stanton number as well as the boundary-layer parameters should be investigated. It is critical that the issue of flow separation in the previous, large Reynolds number computations be clearly addressed.

Finally, a logical continuation of this work would be solution of second-order boundary-layer equations for the same physical geometry and comparison with these computational results. This will
provide a means for quantifying the validity and applicability of the second-order boundary-layer equations for solution of real-world flows.
APPENDIX

Log Sheets for GASP Computational Runs
**SPACE + GLOBAL (PNS + US)**

<table>
<thead>
<tr>
<th>CASE</th>
<th>DATE</th>
<th>REASON FOR RUN</th>
<th>RTOLR</th>
<th>RTOLA</th>
<th>GRID SIZE</th>
<th>RESULTS OF RUN</th>
<th>FILENAMES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3/23</td>
<td>First attempt at run</td>
<td>10^-5</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>3/24</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>3/25</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>3/26</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>3/27</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>3/28</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>3/29</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>3/30</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>3/31</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>4/1</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>4/2</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>4/3</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>4/4</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>4/5</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>4/6</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>4/7</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>4/8</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>4/9</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>4/10</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>4/11</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>4/12</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>4/13</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>4/14</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>4/15</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>4/16</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>4/17</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>4/18</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>4/19</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>4/20</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>4/21</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>4/22</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>4/23</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>33</td>
<td>4/24</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>4/25</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>4/26</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>4/27</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>37</td>
<td>4/28</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>38</td>
<td>4/29</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
<tr>
<td>39</td>
<td>4/30</td>
<td>Run w/ closed curve</td>
<td>10^-4</td>
<td>10^-4</td>
<td>OK</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Results of Run**

- OK: Run successful
- NG: Run unsuccessful
<table>
<thead>
<tr>
<th>#</th>
<th>CASE</th>
<th>DATE</th>
<th>REASON FOR RUN</th>
<th>RTOLR</th>
<th>ATOLA</th>
<th>GRID SIZE</th>
<th>RESULTS OF RUN</th>
<th>FILENAMES</th>
</tr>
</thead>
<tbody>
<tr>
<td>84</td>
<td>2</td>
<td>5125</td>
<td>N1000 Smoothed run</td>
<td>1e-17</td>
<td>1e-17</td>
<td>1e-9 / 1e-7</td>
<td>100</td>
<td>50</td>
</tr>
<tr>
<td>85</td>
<td>4</td>
<td>5125</td>
<td>N1000 Smoothed run</td>
<td>1e-17</td>
<td>1e-17</td>
<td>1e-9 / 1e-7</td>
<td>100</td>
<td>50</td>
</tr>
<tr>
<td>86</td>
<td>4</td>
<td>5125</td>
<td>N1000 Smoothed run</td>
<td>1e-17</td>
<td>1e-17</td>
<td>1e-9 / 1e-7</td>
<td>100</td>
<td>50</td>
</tr>
<tr>
<td>87</td>
<td>4</td>
<td>5125</td>
<td>N1000 Smoothed run</td>
<td>1e-17</td>
<td>1e-17</td>
<td>1e-9 / 1e-7</td>
<td>100</td>
<td>50</td>
</tr>
<tr>
<td>88</td>
<td>4</td>
<td>5125</td>
<td>N1000 Smoothed run</td>
<td>1e-17</td>
<td>1e-17</td>
<td>1e-9 / 1e-7</td>
<td>100</td>
<td>50</td>
</tr>
<tr>
<td>89</td>
<td>4</td>
<td>5125</td>
<td>N1000 Smoothed run</td>
<td>1e-17</td>
<td>1e-17</td>
<td>1e-9 / 1e-7</td>
<td>100</td>
<td>50</td>
</tr>
<tr>
<td>90</td>
<td>4</td>
<td>5125</td>
<td>N1000 Smoothed run</td>
<td>1e-17</td>
<td>1e-17</td>
<td>1e-9 / 1e-7</td>
<td>100</td>
<td>50</td>
</tr>
<tr>
<td>91</td>
<td>4</td>
<td>5125</td>
<td>N1000 Smoothed run</td>
<td>1e-17</td>
<td>1e-17</td>
<td>1e-9 / 1e-7</td>
<td>100</td>
<td>50</td>
</tr>
<tr>
<td>92</td>
<td>4</td>
<td>5125</td>
<td>N1000 Smoothed run</td>
<td>1e-17</td>
<td>1e-17</td>
<td>1e-9 / 1e-7</td>
<td>100</td>
<td>50</td>
</tr>
<tr>
<td>93</td>
<td>4</td>
<td>5125</td>
<td>N1000 Smoothed run</td>
<td>1e-17</td>
<td>1e-17</td>
<td>1e-9 / 1e-7</td>
<td>100</td>
<td>50</td>
</tr>
<tr>
<td>#</td>
<td>CASE</td>
<td>DATE</td>
<td>REASON FOR RUN</td>
<td>RTOLR</td>
<td>RTOLA</td>
<td>GRID SIZE</td>
<td>RESULTS OF RUN</td>
<td>FILENAMES</td>
</tr>
<tr>
<td>---</td>
<td>------</td>
<td>------</td>
<td>----------------</td>
<td>--------</td>
<td>--------</td>
<td>-----------</td>
<td>----------------</td>
<td>-----------</td>
</tr>
<tr>
<td>1</td>
<td>a</td>
<td>10-19</td>
<td>Tests at 2000</td>
<td>1e-10</td>
<td>1e-10</td>
<td>12x12</td>
<td>OK (5x5)</td>
<td>w1.txt</td>
</tr>
<tr>
<td>2</td>
<td>b</td>
<td>12-15</td>
<td>Real Reuse</td>
<td>1e-8</td>
<td>1e-8</td>
<td>16x16</td>
<td>OK (6x6)</td>
<td>w2.txt</td>
</tr>
<tr>
<td>1</td>
<td>c</td>
<td>12-16</td>
<td>Real Real Use</td>
<td>1e-6</td>
<td>1e-6</td>
<td>18x18</td>
<td>OK (7x7)</td>
<td>w3.txt</td>
</tr>
<tr>
<td>2</td>
<td>d</td>
<td>12-17</td>
<td>Test of 2000</td>
<td>1e-4</td>
<td>1e-4</td>
<td>20x20</td>
<td>OK (8x8)</td>
<td>w4.txt</td>
</tr>
<tr>
<td>3</td>
<td>e</td>
<td>12-18</td>
<td>Test of 2000</td>
<td>1e-2</td>
<td>1e-2</td>
<td>22x22</td>
<td>OK (9x9)</td>
<td>w5.txt</td>
</tr>
<tr>
<td>4</td>
<td>f</td>
<td>1113</td>
<td>Same</td>
<td>1e-2</td>
<td>1e-2</td>
<td>24x24</td>
<td>OK (10x10)</td>
<td>w6.txt</td>
</tr>
<tr>
<td>5</td>
<td>g</td>
<td>10-18</td>
<td>Test of 2000</td>
<td>1e-10</td>
<td>1e-10</td>
<td>12x12</td>
<td>OK (5x5)</td>
<td>w7.txt</td>
</tr>
<tr>
<td>6</td>
<td>h</td>
<td>10-19</td>
<td>Test of 2000</td>
<td>1e-10</td>
<td>1e-10</td>
<td>12x12</td>
<td>OK (5x5)</td>
<td>w8.txt</td>
</tr>
<tr>
<td>7</td>
<td>i</td>
<td>12-15</td>
<td>Real Reuse</td>
<td>1e-8</td>
<td>1e-8</td>
<td>16x16</td>
<td>OK (6x6)</td>
<td>w9.txt</td>
</tr>
<tr>
<td>8</td>
<td>j</td>
<td>12-17</td>
<td>Test of 2000</td>
<td>1e-4</td>
<td>1e-4</td>
<td>20x20</td>
<td>OK (8x8)</td>
<td>w10.txt</td>
</tr>
<tr>
<td>9</td>
<td>k</td>
<td>12-18</td>
<td>Test of 2000</td>
<td>1e-2</td>
<td>1e-2</td>
<td>22x22</td>
<td>OK (9x9)</td>
<td>w11.txt</td>
</tr>
<tr>
<td>10</td>
<td>l</td>
<td>1113</td>
<td>Same</td>
<td>1e-2</td>
<td>1e-2</td>
<td>24x24</td>
<td>OK (10x10)</td>
<td>w12.txt</td>
</tr>
<tr>
<td>11</td>
<td>m</td>
<td>10-18</td>
<td>Test of 2000</td>
<td>1e-10</td>
<td>1e-10</td>
<td>12x12</td>
<td>OK (5x5)</td>
<td>w13.txt</td>
</tr>
<tr>
<td>12</td>
<td>n</td>
<td>10-19</td>
<td>Test of 2000</td>
<td>1e-10</td>
<td>1e-10</td>
<td>12x12</td>
<td>OK (5x5)</td>
<td>w14.txt</td>
</tr>
<tr>
<td>13</td>
<td>o</td>
<td>12-15</td>
<td>Real Reuse</td>
<td>1e-8</td>
<td>1e-8</td>
<td>16x16</td>
<td>OK (6x6)</td>
<td>w15.txt</td>
</tr>
<tr>
<td>14</td>
<td>p</td>
<td>12-17</td>
<td>Test of 2000</td>
<td>1e-4</td>
<td>1e-4</td>
<td>20x20</td>
<td>OK (8x8)</td>
<td>w16.txt</td>
</tr>
<tr>
<td>15</td>
<td>q</td>
<td>12-18</td>
<td>Test of 2000</td>
<td>1e-2</td>
<td>1e-2</td>
<td>22x22</td>
<td>OK (9x9)</td>
<td>w17.txt</td>
</tr>
<tr>
<td>16</td>
<td>r</td>
<td>1113</td>
<td>Same</td>
<td>1e-2</td>
<td>1e-2</td>
<td>24x24</td>
<td>OK (10x10)</td>
<td>w18.txt</td>
</tr>
<tr>
<td>17</td>
<td>s</td>
<td>10-18</td>
<td>Test of 2000</td>
<td>1e-10</td>
<td>1e-10</td>
<td>12x12</td>
<td>OK (5x5)</td>
<td>w19.txt</td>
</tr>
<tr>
<td>18</td>
<td>t</td>
<td>10-19</td>
<td>Test of 2000</td>
<td>1e-10</td>
<td>1e-10</td>
<td>12x12</td>
<td>OK (5x5)</td>
<td>w20.txt</td>
</tr>
<tr>
<td>19</td>
<td>u</td>
<td>12-15</td>
<td>Real Reuse</td>
<td>1e-8</td>
<td>1e-8</td>
<td>16x16</td>
<td>OK (6x6)</td>
<td>w21.txt</td>
</tr>
<tr>
<td>20</td>
<td>v</td>
<td>12-17</td>
<td>Test of 2000</td>
<td>1e-4</td>
<td>1e-4</td>
<td>20x20</td>
<td>OK (8x8)</td>
<td>w22.txt</td>
</tr>
<tr>
<td>21</td>
<td>w</td>
<td>12-18</td>
<td>Test of 2000</td>
<td>1e-2</td>
<td>1e-2</td>
<td>22x22</td>
<td>OK (9x9)</td>
<td>w23.txt</td>
</tr>
<tr>
<td>22</td>
<td>x</td>
<td>1113</td>
<td>Same</td>
<td>1e-2</td>
<td>1e-2</td>
<td>24x24</td>
<td>OK (10x10)</td>
<td>w24.txt</td>
</tr>
<tr>
<td>23</td>
<td>y</td>
<td>10-18</td>
<td>Test of 2000</td>
<td>1e-10</td>
<td>1e-10</td>
<td>12x12</td>
<td>OK (5x5)</td>
<td>w25.txt</td>
</tr>
<tr>
<td>24</td>
<td>z</td>
<td>10-19</td>
<td>Test of 2000</td>
<td>1e-10</td>
<td>1e-10</td>
<td>12x12</td>
<td>OK (5x5)</td>
<td>w26.txt</td>
</tr>
</tbody>
</table>
SELECTED BIBLIOGRAPHY


REFERENCES

4 Emanuel, 320.
5 Van Dyke, 288.
6 Emanuel, 320.
8 Emanuel, 317.
9 Ibid., 323.
10 Ibid.
11 Ibid., 326.
12 Ibid., 332.
13 Ibid., 333.
14 Ibid., 332.
17 Ibid, 355.
18 Ibid., 161.
21 Van Dyke, 288.
22 Goldstein, et. al., 231.
24 Sawley, et. al., 135-138.


27 Ibid., 14.

28 Ibid., 15.


30 Ibid., 317.

31 Ibid., 329.

32 Anderson, et. al., 315.


35 McGrory, et. al., 386.


37 GASP Version 2.2, Aerosoft, Inc.

38 Emanuel, 320.


40 Ibid., 2401.

41 Anderson, et. al., 76.

42 Emanuel, 358

43 Hayes, et. al., 339.

44 McGrory, et. al., 161.