Laminar, Steady and Unsteady Flow Over Inclined Plates in Two and Three Dimensions.

by

Evangelos Hytopoulos

Thesis submitted to the Faculty of the Virginia Polytechnic Institute and State University in partial fulfillment of the requirements for the degree of Master of Science in Aerospace Engineering

APPROVED:

Joseph A. Schetz, Chairman

Bernard Grossman

Robert W. Walters

March, 1990

Blacksburg, Virginia
LD
5655
V855
1980
H926
0.2
Laminar, Steady and Unsteady Flow Over
Inclined Plates in Two and Three Dimensions.

by

Evangelos Hytopoulos
Joseph A. Schetz, Chairman
Aerospace Engineering

(ABSTRACT)

The problem studied is the laminar flow over inclined, finite flat plates for moderately high Reynolds numbers in two and three dimensions. There are only few prior analyses, mainly for two dimensional flow, for this problem, and thus it was decided that it was worthwhile to study it now in great detail. The full Navier-Stokes equations were solved using a weak Galerkin formulation for the Finite Element Method with the pressure determined by a penalty approach. The influence of grid resolution, boundary conditions and size of the domain was studied. The true nature of the flow for different Reynolds numbers was also examined through steady and unsteady simulations of the two dimensional cases for $6600 \leq Re_L \leq 18000$. Results for the three dimensional flow over square plates at two angles of attack, $\alpha = 3.0$ and $8.0$ degrees for $Re_L = 100$ are presented. The results are given in terms of skin friction and pressure coefficient variations along with flowfield visualization. Comparison between the two dimensional and three dimensional flow indicates the in-
fluence of the third coordinate to the flow. The analysis indicated that the two dimensional flow over a finite thick plate at 3.0 degrees angle of attack is steady up to Re = 12000. The solution for the upper surface is strongly influenced by the presence of a recirculation bubble at the leading edge. The slope of the lift curve for the 2D viscous flow is less than $2\pi$, the result predicted by the thin wing theory. The solution for the three dimensional flow is strongly influenced by the existence of the tip vortices. The slope of the lift curve for the 3D viscous flow is less than the one corresponding to the 2D flow. In addition, the effect of the aspect ratio on the lift does not agree with the inviscid lifting line theory prediction.
ACKNOWLEDGEMENTS

I would like to express my deep gratitude to my advisor and chairman of the committee Dr. Joseph A. Schetz, who always directed and supported this work with great enthusiasm. I am grateful to him for his encouragement and enlightening teaching, but most of all, for allowing me to feel like his partner in this challenging project.

I would also like to extend my gratitude to Dr. Bernard Grossman and Dr. Robert Walters for their advice and constructive comments on this work.

Special thanks to Jean Caille whose help in understanding and using the Finite Element Method was very important for the successful completion of this work.

Thanks are also extended to William Richardson who, as a consultant in User's Services, was always willing to tackle difficult computational problems, making my job much easier.

Finally, I want to thank my parents, Christos and Anthoula Hytopoulos, whose love, understanding and encouragement provided the foundation of my education.
## TABLE OF CONTENTS

1.0 Introduction ........................................... 1
   1.1 Background ......................................... 1
   1.2 Finite Element Method ............................... 2
   1.3 Study of the flow field over inclined flat plates. 3

2.0 Theoretical background ............................... 8
   Formulation of the continuum and discrete problem .... 8
      2.1 Mathematical Formulation .......................... 8
      2.2 Formulation of the Discrete Problem ............... 12
      2.3 Elements .......................................... 15
      2.4 Solution procedures ................................ 19
      2.5 Transient Algorithms ................................ 21
      2.6 Convergence Criteria ............................... 24
      2.7 Boundary conditions. .............................. 25
      2.8 Software implementation. ........................... 25

3.0 Steady flow over 2D thick flat plates .............. 26
   3.1 Grid choice. ....................................... 26
   3.2 Boundary conditions. ................................ 28
   3.3 Results. ........................................... 30
   3.4 Discussion of the results. .......................... 42

Table of Contents v
4.0 Unsteady flow over 2D thick flat plate ........ 45
4.1 Boundary and initial conditions ................. 45
4.2 Results ........................................... 46
4.3 Discussion ......................................... 49

5.0 Results for steady flow over 3D flat plates .... 51
5.1 Problem choice ..................................... 51
5.2 Grid choice ......................................... 52
5.3 Boundary conditions ............................... 55
5.4 Solution strategy ................................... 55
5.5 Results ............................................. 56

CONCLUSIONS ............................................. 63

FIGURES .................................................. 66

References .............................................. 120

Vita ..................................................... 123

Table of Contents vi
**LIST OF ILLUSTRATIONS**

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Quadrilateral and brick elements.</td>
<td>67</td>
</tr>
<tr>
<td>2.</td>
<td>&quot;C-type&quot; grid.</td>
<td>68</td>
</tr>
<tr>
<td>3.</td>
<td>Geometry of the thick plate.</td>
<td>69</td>
</tr>
<tr>
<td>4.</td>
<td>&quot;H-type&quot; grid.</td>
<td>70</td>
</tr>
<tr>
<td>5.</td>
<td>Skin friction coefficient for $Re = 10000$, $a=3.0$ degrees.</td>
<td>71</td>
</tr>
<tr>
<td>6.</td>
<td>Skin friction and pressure coefficients for $Re=10000$, $a=3.0$ deg.</td>
<td>72</td>
</tr>
<tr>
<td>7.</td>
<td>Domain independent solution for $Re=10000$, $a=3.0$ deg.</td>
<td>73</td>
</tr>
<tr>
<td>8.</td>
<td>Skin friction and pressure coefficients for $Re=10000$, $a=3.$ deg.</td>
<td>74</td>
</tr>
<tr>
<td>9.</td>
<td>Skin friction coefficient for thick and thin plates.</td>
<td>75</td>
</tr>
<tr>
<td>10.</td>
<td>Pressure coefficient for thick and thin plates.</td>
<td>76</td>
</tr>
<tr>
<td>11.</td>
<td>$Re=10000$, $a=3.0$ degrees.</td>
<td>77</td>
</tr>
<tr>
<td>12.</td>
<td>Streamline pattern around the leading edge.</td>
<td>78</td>
</tr>
<tr>
<td>13.</td>
<td>Skin friction and pressure coefficients for thick plates.</td>
<td>79</td>
</tr>
<tr>
<td>14.</td>
<td>Grid independent solution for $Re=8000$, $a=3.0$ degrees.</td>
<td>80</td>
</tr>
<tr>
<td>15.</td>
<td>Velocity profiles at the midchord and trailing edge.</td>
<td>81</td>
</tr>
<tr>
<td>16.</td>
<td>Skin friction and pressure coefficient for $Re=12000$, $a=3.0$ degrees.</td>
<td>82</td>
</tr>
<tr>
<td>17.</td>
<td>$Re=12000$., $a=3.0$ degrees.</td>
<td>83</td>
</tr>
<tr>
<td>18.</td>
<td>Skin friction coefficient for different first element height.</td>
<td>84</td>
</tr>
</tbody>
</table>

List of Illustrations vii
Figure 19. Skin friction and pressure coefficients for Re=15000., a=3.0 deg. ............... 85

Figure 20. Skin friction coefficient for Re=15000., a=3.0 degrees. ....................... 86

Figure 21. Re=15000., a = 3.0 degrees, Skin friction coefficient. ......................... 87

Figure 22. Apparent effect of the height of the first element on the solution. .............. 88

Figure 23. Comparison between two solutions for Re = 18000. ............................. 89

Figure 24. Apparent effect of the height of first element. ............................... 90

Figure 25. Behavior of the residuals corresponding to the solution of different Re numbers. ....... 91

Figure 26. Skin friction and pressure coefficients vs. Re. ................................. 92

Figure 27. Local values of skin friction coefficient along the plate vs. Re. ................. 93

Figure 28. Recirculation bubble length vs. Re. Thick plates. ............................. 94

Figure 29. Steady state for the unsteady simulation starting at Re=6600 and ending at Re=10000. 95

Figure 30. Vortex formation and growth at the leading edge. Thick plate. ............... 96

Figure 31. Steady vs. unsteady simulation for Re = 8000. ................................ 97

Figure 32. Steady vs. unsteady simulation for Re=12000., t=3.35. .......................... 98

Figure 33. Steady vs. unsteady simulation for Re = 12000, t=5.39. ........................ 99

Figure 34. Skin friction coefficient vs. x for Re = 18000. ............................... 100

Figure 35. Skin friction coefficient for the region after the recirculation zone. ........... 101

Figure 36. 3D mesh. ......................... 102

List of Illustrations viii
Figure 37. Skin friction and pressure coefficient for 2D flow. ........................................ 103

Figure 38. Pressure coefficient for the centerplane. ......................................................... 104

Figure 39. Vortex at the side edge of the plate. Re=100, a=3.0 degrees. ............................ 105

Figure 40. Vortex formation for the two angles of attack. .................................................. 106

Figure 41. Top view of the flow field at the side edge of the plate. ................................... 107

Figure 42. Pressure field. Upper and lower surfaces of the plate. ...................................... 108

Figure 43. Pressure field. Lower surface for a = 3.0 and 8.0 degrees. .................................. 109

Figure 44. Vorticity field 0.0001*L above the plate. .......................................................... 110

Figure 45. Vorticity field, centerplane and plane at the side edge. ..................................... 111

Figure 46. Skin friction. 2D versus 3D calculations (centerplane). .................................... 112

Figure 47. Pressure coefficient. 2D vs 3D calculations (centerplane). ................................ 113

Figure 48. Lift production, expressed in terms of pressure difference. ............................... 114

Figure 49. Re=100, a=3.0 and 8.0 degrees. ................................................................. 115

Figure 50. Skin friction coefficient on the centerplane, 3D flow. ..................................... 116

Figure 51. W-component of the velocity along the side edge. ............................................ 117

Figure 52. Velocity profiles along the trailing edge. .......................................................... 118

Figure 53. Re=100., a=8.0 degrees. ................................................................................. 119
1.0 INTRODUCTION

1.1 Background

Historically, the time-proven approach to engineering design has been to construct a physical model and conduct laboratory experiments to "measure the data" affecting the design decision. In recent years, major advances in computer technology and computational mechanics have made it possible to construct numerical models and again "measure the data" obtained from the computational "experiment". This computational approach allows the engineer to investigate different model configurations under different physical conditions in a relatively short time and at a greatly reduced cost. However, computational and experimental techniques are complementary and not mutually exclusive approaches to an engineering problem. Although numerical simulation does not eliminate the need for building and testing prototypes, it can considerably reduce the number and variety of such tests required, as well as enhance the effectiveness of testing.

Also, some situations occur where a numerical solution is practical and an experiment is not. One example is the fine details of the flow in the case of a thin plate at modest Reynolds numbers where the physical dimensions of a physical
experiment are too small to probe. There are numerous others.

1.2 Finite Element Method

The specific numerical solution method used in this thesis is based on the Finite Element Method (FEM) [1-3], a technique which, although used for some time in structural problems, has a relatively short history in Computational Fluid Dynamics. In recent years, however, intense research has shown that FEM is becoming as powerful a tool in fluid mechanics as it is in structural analysis. It has been especially successful in simulating many classes of incompressible fluid flows. In FEM, the flow region is subdivided into a number of small regions, called finite elements. The dependent variables are interpolated within each element by functions of compatible order. Using these approximations in the field equations, we introduce residuals or errors. The Galerkin method seeks to reduce these errors to zero in a weighted sense and thus, the partial differential equations describing the problem in the region as a whole are replaced by ordinary differential equations or algebraic equations in each element. This system of equations
is then solved simultaneously by numerical techniques to determine the velocities and pressures in the region.

Some of the advantages of FEM over other methods such as finite difference methods (FDM) are its inherent flexibility in treating complex geometrical domains and boundary conditions. Unstructured grids can be easily used which allows regions of interest to be studied in detail without the need for excessively many grid points throughout the entire domain. The Finite Element Method allows the natural and correct imposition of boundary conditions on curved boundaries. The natural traction-free boundary condition is especially useful for external flows. In addition, the elegant mathematical formulation of FEM allows the derivation of comprehensive error estimates and the determination of accurate solutions to within user-prescribed tolerances.

1.3 Study of the flow field over inclined flat plates.

As the title of this thesis indicates, the aim is to obtain numerical solutions of high accuracy for the flow field over finite flat plates at an angle of attack for various Reynolds numbers. The simple geometry of a flat plate allows focusing on some important characteristics of the flow field. This flow is an idealized representation of many flows of
practical interest such as flows over propeller blades and finite, thin wings. The study includes both steady and time-consistent unsteady calculations for 2D and 3D laminar flow.

The main objective was to conduct numerical "experiments" to study the flow fields in great detail. The secondary objectives were to study the influence of the grid resolution, sensitivity of the solution to the boundary conditions and the behavior of the solution when the steady and unsteady algorithms are used.

Since there were only few prior analyses of the problem in the literature, we decided that it was worthwhile to revisit the problem. In a paper published in 1927 [4], Fage and Johansen presented experimental data for different angles of attack with emphasis on the presentation of the results concerning the vortex system behind the flat plate. More recently, Lught and Haussling [5] obtained numerical solutions by means of a stream function-vorticity formulation for the laminar incompressible flow past a 2D flat plate for angles of 0, 45 and 90 degrees and \( \text{Re} = 30, 50, \) and 200, using the finite difference method. The initial-boundary value problem for the Navier-Stokes equations was solved in an elliptic coordinate system. Potential flow was selected as the initial condition, and the flow was assumed to be two-dimensional and time-dependent. Also, Schmall and Kinney

1.0 Introduction
[6], have performed a numerical investigation of the two-dimensional problem for an angle of attack of 30 degrees and Re = 4 and 400. The problem was analyzed from the standpoint of vorticity production and transport using the complete vorticity transport equation. The entire fluid motion was ultimately calculated through application of the velocity induction law (Law of Biot-Savart). Although this relationship is most familiar to those dealing with inviscid flow, since it is purely kinematical, it is applicable to viscous flows as well.

Some numerical investigation of the problem for compressible flows is presented in [7] by S.V.Ramakrishnan and S.G.Rubin. In their paper, they treated the 2D flow over a flat plate without thickness for \( M_\infty = 0.3 \) and different combinations of angle of attack and Re. Their numerical solutions are primarily obtained with a reduced form (RNS) of the Navier-Stokes equations. First order forward differencing is used to approximate the temporal derivatives; the convective streamwise \((x)\) derivatives are approximated by a first order implicit backward differencing formula. In separated regions, the streamwise convection term in the streamwise momentum equation is approximated by a first order upwind formula. Results for steady and time-consistent, unsteady simulations are presented. The influence of grid resolution is discussed, especially for the regions near the

1.0 Introduction
leading and trailing edges. Some discussion about the stability of laminar flow for larger Re calculations is also presented.

Caille and Schetz [16] have performed 2D and 3D numerical simulations of the flow field around finite flat plates at zero angle of attack. The full Navier-Stokes equations were solved using the weak Galerkin formulation for the Finite Element Method with the pressure determined by a penalty approach. The results are presented mainly in terms of the skin friction coefficient on the plate, the wake centerline velocity variation and the velocity variation along the upstream stagnation streamline. The results are compared to earlier analyses and experiments that considered various individual parts of the flow when available.

Since the results presented in this thesis cover a broad range of flow conditions, a systematic presentation is required. In chapter 3, the 2D steady flow over inclined flat plates is presented for different Re. The influence of thickness is studied. The results are mainly given in terms of the skin friction and pressure coefficients. Factors that influence the convergence and grid-independence of the solution are also discussed. In Chapter 4, the results for time-consistent, 2D simulations are presented. Investigation of the 2D flow cases, treated in the previous chapter as steady state cases, are treated as unsteady revealing some
very interesting features of the flow especially at the higher values of Re. Chapter 5 is devoted to the presentation of the results concerning the 3D flow simulations. Comparisons with 2D cases are included to show the effect of the third coordinate, mainly in terms of skin friction and pressure coefficient. Interesting figures showing the development of the vortex and the velocity profiles in the wake and the side edge of the plate are also used to visualize the flow field.
2.0 THEORETICAL BACKGROUND

FORMULATION OF THE CONTINUUM AND DISCRETE PROBLEM

2.1 Mathematical Formulation

Application of mass conservation results in the continuity equation,

$$\frac{\partial \rho}{\partial t} + (\rho u_j)_j = 0$$  \hspace{1cm} [2.1.1]

From the balance of linear momentum with surface and body forces, we obtain

$$\rho \left( \frac{\partial u_i}{\partial t} + u_j u_i, j \right) = \sigma_{ij}, j + \rho f_i$$  \hspace{1cm} [2.1.2]

where:

$$\sigma_{ij} = -\rho \delta_{ij} + t_{ij}$$  \hspace{1cm} [2.1.3]

The boundary conditions are specified by a combination of prescribed velocities and stresses. Thus, if \( \Gamma \) is the boundary of the fluid volume \( \Omega \) with the different portions denoted, \( \Gamma_u, \Gamma_t \), the boundary conditions are:

$$u_i = \bar{u}_i(s, t) \text{ on } \Gamma_u$$  \hspace{1cm} [2.1.4]
\[ \sigma_i = \sigma_{ij} n_j(s) = \mathbf{f}_i(s, t) \quad \text{on} \quad \Gamma_t \]  

[2.1.5]

where \( s \) is the length along the boundary, and \( n_j(s) \) is the outward normal.

The initial conditions have the form

\[ u_i = u_i^0(x, 0) \quad x \in \Omega \]  

[2.1.6]

Equations (2.1.1)-(2.1.6) define the fluid flow in its most general form. In order to make these equations tractable for numerical simulation, a number of simplifying assumptions can be made. The assumptions made here are:

(1) The density is constant

(2) The constitutive relation is assumed to be:

\[ \tau_{ij} = 2\mu \varepsilon_{ij} \]

\[ \mu = \mu(T) = \text{const.} \quad \text{since we assume no temperature variation} \]

(3) The fluid motion is laminar

(4) The boundary conditions apply at fixed boundaries.

With these assumptions, equations (2.1.1) - (2.1.6) reduce to:

\[ u_i, t = 0 \]  

[2.1.7]

2.0 Theoretical background
\[
\rho_0 \left( \frac{\partial u_i}{\partial t} + u_j u_i, j \right) = \sigma_{ij}, j + \rho_0 \dot{e}_i
\]  \[2.1.8\]

\[
\sigma_{ij} = -p \delta_{ij} + 2 \mu \varepsilon_{ij}
\]  \[2.1.9\]

where:

\[
\varepsilon_{ij} = \frac{1}{2} \left( u_{i,j} + u_{j,i} \right)
\]  \[2.1.10\]

Then introducing (2.1.10) into (2.1.9) and dropping the subscript on the density one gets

\[
\rho \left( \frac{\partial u_i}{\partial t} + u_j u_i, j \right) = -p, i + \rho f_i + \left[ \mu (u_{i,j} + u_{j,i}) \right], j
\]  \[2.1.11\]

This form of the momentum equation is sometimes known as the stress-divergence form. In the case that the viscosity is constant, the momentum equation can be written as:

\[
\rho \left( \frac{\partial u}{\partial t} + u \cdot \nabla u \right) = -\nabla p + \rho f + \mu \nabla^2 u
\]  \[2.1.12\]

The form (2.1.12) is known as the Navier-Stokes form of the momentum equation.

Often the FEM method is not applied directly to the system of equations (2.1.8) - (2.1.12) but rather to a perturbed system of equations in which the continuity requirement is weakened and replaced by,

2.0 Theoretical background
\[ u_{i, i} = -\varepsilon p \]  \hspace{1cm} [2.1.13]

where \( \varepsilon \), the penalty parameter [9], is small \((10^{-5} - 10^{-9})\).

It can be shown (see Bercovier [10]), if \((u^s, p^s)\) is the solution of the perturbed system, then under certain conditions [11],

\[ \|u - u^s\|_1 + \|p - p^s\|_0 \leq C\varepsilon ; \quad C = C(\Omega, f) \]  \hspace{1cm} [2.1.14]

where the \textbf{1-norm}, \( \| \cdot \|_1 \) takes into account the inner product of the derivatives of the solution and the \textbf{zero-norm} is the \( L^2 \) norm. Thus, one can solve the perturbed system in place of the original system without losing any significant accuracy, provided \( \varepsilon \) is small enough. This procedure has the advantage of eliminating the dependent variable \( p^s \), which is recovered by post-processing from the velocity field by,

\[ p^s = -\frac{1}{\varepsilon} u_{i, i} \]  \hspace{1cm} [2.1.15]
2.2 Formulation of the Discrete Problem.

The objective of the Finite Element Method is to reduce the continuous problem of the governing equations to a discrete problem described by a system of algebraic equations. The procedure begins with the division of the continuum region of interest into a number of simply shaped regions called elements. Within each element, the dependent variables are interpolated by functions of compatible order in terms of values to be determined at a set of nodal points. This is an essential difference between finite differences and FEM. In the first method, one approximates the derivatives, and, in the second, one approximates the solution. In order to develop the equations for these nodal point unknowns, an individual element may be separated from the assembled system. Within each element, the velocity and the pressure fields are approximated by,

\[ u_i(x, t) = \phi^T U_i(t) \]

\[ p(x, t) = \psi^T P(t) \]  \[2.2.16\]

where \( U_i \) and \( P \) are column vectors of element nodal point unknowns and \( \phi, \psi \) are column vectors of the interpolation functions. Introducing these approximations in the field equations one gets a set of equations:

2.0 Theoretical background
\[ f_1(\phi, \psi, U_i, P) = R_1 \quad \text{Momentum} \]

\[ f_2(\phi, U_i) = R_2 \quad \text{Incompressibility} \quad \text{[2.2.17]} \]

where \( R_1, R_2 \) are residuals resulting from the use of the approximations.

The Galerkin form of the method of Weighted Residuals seeks to reduce these errors to zero, in a weighted sense, by making the residuals orthogonal to the interpolation functions of each element. These orthogonality conditions are expressed by,

\[ (f_1, \phi) = (R_1, \phi) = 0 \]

\[ (f_2, \psi) = (R_2, \psi) = 0 \quad \text{[2.2.18]} \]

where \((a,b)\) denotes the inner product, defined by,

\[ (a,b) = \int_V a \cdot b \, dV \quad V \text{ being the volume of the element. The results of those computations are expressed by the following matrix equations:} \]

\[ \text{Momentum} \quad M \dot{U} + A(U)U + K(U)U - CP = F \quad \text{[2.2.19]} \]

\[ \text{Incompressibility} \quad C^T \dot{U} = 0 \quad \text{[2.2.20]} \]

2.0 Theoretical background
with \( U = (U_1, U_2, U_3) \). Now, one can write this system in the following form introducing the vector of unknowns \( V = (U_1, U_2, U_3, P)^T \)

\[
\begin{bmatrix}
M & 0 \\
0 & 0
\end{bmatrix}
\begin{bmatrix}
\dot{U} \\
P
\end{bmatrix}
+ \begin{bmatrix}
A(U) + K(U) & -C \\
-C^T & 0
\end{bmatrix}
\begin{bmatrix}
U \\
P
\end{bmatrix}
= \begin{bmatrix}
F \\
0
\end{bmatrix}
\]  \hspace{1cm} [2.2.21]

The above form corresponds to the Mixed Formulation. When the penalty formulation is employed, the incompressibility condition is replaced by the discrete analogue of equation (2.1.13); i.e.,

\[ C^T U = -\varepsilon M_p P \]  \hspace{1cm} [2.2.22]

This equation is used to eliminate the pressure from the momentum equation giving the matrix form:

\[ M\ddot{U} + A(U)U + K(U)U + \frac{1}{\varepsilon} C M_p^{-1} C^T U = F \]  \hspace{1cm} [2.2.23]

Obviously, the penalty formulation reduces the unknowns by one, which results in reduced memory requirements. The pressure can be recovered after the calculation of the velocity field. In the previously defined matrix forms, \( A \) represents the advection of the momentum; \( K \) represents the diffusion of momentum; \( M \) represents the mass terms in the field equations, and the \( F \) vector provides the forcing func-

2.0 Theoretical background
tions for the system in terms of body forces and surface traction.

The discrete representation of the entire region of interest is obtained through an assembly procedure over the total number of elements in the domain where inter-element continuity of velocity and equilibrium of internal and external forces is enforced. The result of such an assembly is a system of matrix equations of the form given by equation (2.2.21) or equation (2.2.23)

2.3 Elements

One of the most important aspects in FEM is the choice of the particular elements used for the simulation. Elements for fluid flow are usually categorized by the combinations of the velocity-pressure approximation used. Many different velocity-pressure combinations are available in the literature, and some combinations are more efficient than others taking into account the degree of interpolation, the memory required, the number of elements required for an acceptable solution, etc. Two different types of elements, for the 2D and 3D, cases were used in this study. For the 2-D cases the 9 node isoparametric quadrilateral was used, with biquadratic interpolation for the velocity and a discontinuous linear pressure approximation. For the 3-D cases, due to memory...
limitations, the 8 node isoparametric brick with trilinear interpolation of the velocity and discontinuous constant pressure approximation was used.

a) 9 node quadrilateral: For this element, the velocity is approximated using biquadratic interpolation functions, \( \phi \in Q_2 \), given by

\[
\phi = \begin{bmatrix}
\frac{1}{4} rs(1 - r)(1 - s) \\
- \frac{1}{4} rs(1 + r)(1 - s) \\
\frac{1}{4} rs(1 + r)(1 + s) \\
- \frac{1}{4} rs(1 - r)(1 + s) \\
- \frac{1}{2} s(1 - s)(1 - r^2) \\
\frac{1}{4} r(1 + r)(1 - s^2) \\
\frac{1}{2} s(1 + s)(1 - r^2) \\
- \frac{1}{2} r(1 - r)(1 - s^2) \\
(1 - r^2)(1 - s^2)
\end{bmatrix}
\]

[2.3.24]

The discontinuous approximation was chosen for the pressure. Since the penalty method was used, the pressure is evaluated after the solution for the velocity field has been obtained at the four points of 2\times2 Gaussian integration.

2.0 Theoretical background
b) 8 node brick element: For this element, the velocity components \( u_i \) are approximated using trilinear interpolation functions, i.e. \( \phi \in Q_1 \). These are defined by,

\[
\phi = \begin{bmatrix}
\frac{1}{8} (1-r)(1-s)(1-t) \\
\frac{1}{8} (1+r)(1-s)(1-t) \\
\frac{1}{8} (1-r)(1+s)(1-t) \\
\frac{1}{8} (1+r)(1+s)(1-t) \\
\frac{1}{8} (1-r)(1-s)(1+t) \\
\frac{1}{8} (1+r)(1-s)(1+t) \\
\frac{1}{8} (1-r)(1+s)(1+t) \\
\frac{1}{8} (1+r)(1+s)(1+t)
\end{bmatrix}
\]  

[2.3.25]

For the pressure, a piecewise constant discontinuous approximation, \( \psi \in Q_0 \) was used. The pressure degree of freedom in this case is usually associated with the element centroid.

A close look at the governing equations reveals that the pressure behaves like the derivative of the velocity. Thus, for all the elements described above and for most of the elements used for fluid flows the pressure interpolation functions are one degree less than the velocity approximation functions. More rigorously: for the Navier-Stokes equations, the inclusions \( V_h^0 \subset H_0^1(\Omega) \land S_h^0 \subset L^2_0(\Omega) \), where \( V_h^0 \) and \( S_h^0 \) are the finite element spaces for the velocity and

2.0 Theoretical background
pressure respectively, are not by themselves sufficient to produce stable, meaningful approximations. So, for mixed finite element methods a number of conditions should be satisfied by the finite element spaces. Although most of these are satisfied by arbitrary choices of conforming finite element spaces, one of the conditions restricts this choice. This is the Ladyzhenskaya–Babuska–Brezzi or the inf-sup condition. Loosely speaking, this condition ensures that as the characteristic dimension of the element $h \to 0$, discretely solenoidal functions tend to solenoidal functions. In practice, this condition is easier to satisfy when the interpolation functions used for the pressure are one degree less than those used for the velocity.

The interpolation functions are expressed in terms of the normalized or natural coordinates for the element, $r$, $s$ and $t$, which vary from $-1$ to $+1$ (see Fig. 1). The correspondence between the physical coordinates $(x,y,z)$ and the natural $(r, s, t)$, is defined using a parametric transformation,

$$
\begin{align*}
x &= N^T x \\
y &= N^T y \\
N^T &= N^T(r, s, t)
\end{align*}
$$

[2.3.26]

where $N$ is a vector of interpolation functions over the element and $x,y,z$ are vectors of coordinates describing the geometry of the element. The transformation defined by equation (2.3.26) is quite general and for the 9 node

2.0 Theoretical background

18
quadrilateral allows for the generation of curved-sided elements. If \( N = \phi \); i.e., the interpolation functions defining the dependent variable are of the same order as the functions defining the element geometry, the element is called isoparametric.

2.4 Solution procedures

Typically, the solution phase of a steady-state, Navier-Stokes FEM simulation represents the most time-consuming stage of the analysis. Therefore, the decision as to which solution algorithm to use for this phase can govern and ultimately limit the size of the finite element model that can be treated.

2.4.1) Fixed point Iteration

This is a particularly simple scheme, known also as successive substitution (SS). Here, the nonlinearity is evaluated at the known iterate \( U_i \), and a nonsymmetric linear system must be formed and solved at each iteration. Although the rate of convergence of this iterative procedure is only asymptotically linear, the method is robust and is relatively insensitive to the initial guess, making the method suitable for the first one or two iterations.

2.0 Theoretical background
2.4.2) Newton-Raphson (NR) method

This is a well known method for solving systems of equations \( R(u) = 0 \). Expanding \( R \) into a Taylor series and keeping the terms up to the linear one, we devise an iterative procedure which can be written as follows:

\[
J(u_i)\Delta u_i = -R(u_i) \quad ; \quad u_{i+1} = u_i + \Delta u_i \tag{2.4.27}
\]

where \( J(u) \) is the Jacobian matrix of the system of equations. This method has a rate of convergence superior to the SS. The convergence is asymptotically quadratic, as long as the initial solution vector is within the radius of convergence. Because this is not the case for most problems, it is generally difficult to start the solution procedure using this method. Simultaneously, NR suffers from the same computational drawback as the SS - the construction and complete factorization of a non-symmetric matrix is required at each iterative cycle.

2.4.3) Quasi Newton (QN) Updates

These methods [12-14] are a compromise between the NR method, where the Jacobian is recomputed at every iteration, and the modified Newton method (MNR) where the Jacobian is left unchanged. They derive from the idea of updating the inverse of the Jacobian matrix in simple manner at each iteration rather than recomputing it entirely (NR) or leave it
unchanged (MNR). In practice, the convergence rate of the quasi-Newton algorithm often approaches that of the NR method.

Since none of the above algorithms have the required behavior of convergence, a strategy involving the use of combinations of the available iterative solution methods has to be followed. The strategy uses initially 2 to 3 SS, which has a larger radius of convergence than the NR methods, in order to bring the solution vector within the radius of convergence of the NR method or the QN. The higher convergence rate of these methods can be then exploited. As a first guess for the solution vector, either the Stokes solution or the solution for a lower Re is used.

2.5 Transient Algorithms

The equation

\[ \bar{M} \frac{dU}{dt} + \bar{K}(U)U = \bar{F} \]  \[2.5.28 \]

represents a discrete-space, continuous-time approximation (semidiscrete) to the field problem. A direct time integration replaces the continuous time derivative with an approximation for the history of the dependent variable over a

2.0 Theoretical background
small portion of the problem time scale. The result is an incremental procedure that advances the solution by discrete steps in time, thus using a FEM for the spatial part of the equations and a FDM for the time dependent part.

In constructing such a procedure, questions of stability and accuracy must be considered. Here it should be emphasized that if a penalty formulation is employed, the presence of the penalty parameter in equation (2.5.28) makes it almost mandatory to use an implicit integration scheme. If an explicit scheme were to be used the large order of magnitude of $\varepsilon^{-1}$ would force extremely small time increments for stability to be ensured.

A second-order scheme, a trapezoidal rule, with a variable time step determined by control of the local truncation error was the one used for the simulation of time-dependent problems [15]. It would, of course, be desirable to be able to estimate and control the global error but, unfortunately, there is little or no theory to guide us. The theory does give the result that a local truncation error of $O(dt^3)$ gives a global error which does go like $dt^2$ [19]. The predictor step is obtained by the variable step Adams-Bashforth (AB) formula

$$U_{n+1}^p = U_n + \frac{dt_n}{2} \left[ \left( 2 + \frac{dt_n}{dt_{n-1}} \right) \dot{U}_n - \frac{dt_n}{dt_{n-1}} \dot{U}_{n-1} \right] \quad [2.5.29]$$

2.0 Theoretical background
The corrector step uses the non-dissipative, completely stable trapezoid rule (TR) which applied to (2.5.28) is

\[ \bar{M} \frac{U_{n+1}^c - U_n}{\Delta t_n} + \frac{1}{2} [\bar{K}(U_{n+1}^c)U_{n+1}^c + \bar{K}(U_n)U_n] = \frac{1}{2} [\bar{F}_{n+1} + \bar{F}_n] \tag{2.5.30} \]

The acceleration vector is updated each step by "inverting" the TR in the form

\[ \dot{U}_{n+1} = \frac{2}{\Delta t_n} (U_{n+1} - U_n) - \dot{U}_n \tag{2.5.31} \]

where \( U_n \) is available from the previous application of the same equation.

In order to get the solution at each time step, a nonlinear system of algebraic equations must be solved. This system of nonlinear equations is identical in form to the matrix equation for steady-state problems, and all the methods previously described can be used to solve the nonlinear system of equations that arises for \( U_{n+1} \). In the finite element context, any velocity field that satisfies the discrete analogue of the incompressibility equation can be used as initial condition for the velocity field. No initial conditions are required for the pressure, since there will be a unique pressure field accommodating any solenoidal velocity field.

2.0 Theoretical background
2.6 Convergence Criteria

If a solution based on a particular nonlinear iterative method is to be effective, appropriate criteria should be used to terminate the iteration. A combination of two such criteria to check the convergence of each of the cases was used. The first one measures the relative error in the solution vector $u_i$, and the second one is a measure of the error in the residual vector.

For the solution vector $u_i$, it is required that

$$\|\delta u_i\| \leq \varepsilon_u ; \quad \delta u_i = u_i - u$$

[2.6.32]

where $\| \cdot \|$ is an appropriate norm - usually the Euclidean norm and $\varepsilon_u$ is a prescribed tolerance. Since $u$ is not known a priori and must be approximated, an obvious choice is $\|u_i\|$ for $\|u\|$ and $u_{i-1}$ for $u$ in $\delta u_i = u_i - u_{i-1}$.

For the residual vector we require, that

$$\frac{\|R(u_i)\|}{\|R_0\|} \leq \varepsilon_r$$

[2.6.33]

where $R_0$ is a reference vector, typically $R(U_0)$. Experience [8] has shown that combination of these criteria provides an effective overall convergence for all cases.

2.0 Theoretical background
2.7 Boundary conditions.

In FEM the boundary conditions are specified by a combination of prescribed velocities and stresses. The total stresses appear naturally as boundary conditions during the application of the Galerkin form of the method of Weighted Residuals. Using the Gauss theorem, the term \( \int \sigma_{ij} \cdot \psi \, dV \) (see eq. 2.1.2), can be written as:

\[
\int \sigma_{ij} \cdot \psi \, dV = \int \sigma_{ij} n_j \psi \, dS - \int \sigma_{iij} \cdot \psi \, dV
\]

The term \( t_i = \sigma_{ij} n_j \) appearing in the surface integral is the \( i \)-th traction component on the boundary. Very often, the so-called "traction-free" conditions, i.e. \( t_i = 0 \), are used allowing unconstrained inflow and outflow through the outer boundaries and are much less restrictive on the flow than other conditions. This allows use of smaller computational regions [16].

2.8 Software implementation.

Taking into consideration the specific requirements of the problems addressed in this work, it was decided that the adoption and adaptation of the well-tested FEM computer code FIDAP [8] was preferable to developing a new code. This code was chosen, since it meets all of our desired needs.
3.0 **STEADY FLOW OVER 2D THICK FLAT PLATES**

3.1 **Grid choice.**

In this work, two types of grids were used. For cases where thickness was taken into account, a C-type grid was used (see Fig. 2). As can be seen, the external boundary is rectangular, because this shape is more convenient for the imposition of boundary conditions. Regions where more nodes are concentrated are the leading and trailing edge of the plate and also the region where the geometry of the plate changes. Nodes were also concentrated in the normal to the plate direction for accurately describing the steep gradients in the boundary layer. The distribution of the nodes in this direction is described by a geometrical progression, where the total height of the region ($H$), the height of the first element ($h_1$) and the coefficient of the geometrical progression ($a$) are related as follows:

$$H = h_1 \times \frac{(a)^{n-1} - 1}{(a) - 1} \quad [3.1.1]$$

The ratios used were between 1.05 and 1.25 depending on the total height of the region, the height of the first element, and the specific nature of the flow.
The grid away from the plate was constructed in such a way that the ratio of the faces of the elements is reasonable (~ 10). This is crucial for the accuracy of the finite element solution [10]. Frequently, a higher ratio has been used in those regions where the gradients are not steep with no problem of convergence or accuracy.

The geometry of the plate with thickness treated is presented in Fig. 3. For the leading edge, both continuity of the first derivative between the curved nose and the straight line describing the plate and continuity of the curvature was sought. Unfortunately, the elements used do not allow the satisfaction of this requirement, since they are described by second order polynomials in contrast to the third order required for satisfaction of all the geometric restrictions that are mentioned above. For this reason, the continuity of the first derivative was enforced, and the curvature was kept as small as possible.

The second type of grid, used for the 2D cases where thickness was not taken into account as well as the one used for the 3D calculations, was an H-type grid (Fig. 4). The distribution of the nodes was as described before. This grid is easier to construct, but a lot of attention is needed for the leading and trailing edges which may behave as singular points due to the lack of thickness.

3.0 Steady flow over 2D thick flat plates
The strategy used for obtaining a grid-independent solution for each Re was to first investigate the influence of the location of the external boundaries and then study the influence of the height of the first element used on the surface of the plate. Numerical simulation of the flow over a flat plate at zero angle of attack [16] indicated that a distance of 2 to 4 times the thickness of the boundary layer for the location of the external boundary gave grid-independent solutions. In this study, the location of the upstream, upper and lower boundaries of the domain was varied, and, at the same time the downstream boundary was kept constant at a location equal to two times the length of the plate from the trailing edge. The validity of that choice was checked by comparing the solution obtained with the downstream boundary located at a position equal to two and three times the length of the plate respectively.

3.2 Boundary conditions.

For convenience, define the boundaries of our numerical domain as follows:

- upstream boundary denoted by the letter (W)
- downstream boundary denoted by the letter (E)
o upper boundary denoted by the letter (N)

o lower boundary denoted by the letter (S)

Expanding this notation to three dimensions for later use as:

o right boundary denoted by the letter (R)

o centerplane (CP)

o left boundary denoted by the letter (L)

Since we are dealing with flat plates at an angle of attack, two obvious sets of boundary conditions for the outer boundaries are:

a) Define the essential boundary conditions on the (W) and (S) boundaries (i.e., \( U_\infty, V_\infty \), corresponding to the given angle of attack), and the traction free conditions on the (N) and (E) boundaries. In this case the plate is horizontal and the boundary conditions are inclined.

b) Define the essential boundary conditions on the inflow boundary (W) (i.e., \( U_\infty = 1.0, V_\infty = 0.0 \)), and traction free conditions on the rest of the boundaries. In this case, the plate is at an angle of attack and the approach velocity is horizontal.

3.0 Steady flow over 2D thick flat plates
Numerical experiments with the first set of boundary conditions quickly indicated difficulties getting grid-independent solutions. The solutions obtained for different locations of the external boundaries were different from each other and from the ones obtained using the second set of boundary conditions. It was also obvious that a bigger numerical domain was required, because this kind of boundary conditions was more constraining on the flow than the traction-free conditions. Finally, the region where recirculation occurred was very sensitive to the location of the external boundary. Due to this behavior, the first set of conditions was abandoned, and the second set of boundary conditions was exclusively used for all the calculations.

3.3 Results.

The results presented here are for an angle of attack of 3.0 degrees and Re: 8000, 10000, 12000, 15000 and 18000. To check the grid-independence of the results, the skin friction coefficient and the pressure coefficient were plotted, since the derivatives of the solution converge slower than the solution itself for a given characteristic dimension of the element.
The first test case was $Re = 10000$ and thus although it is not the lowest $Re$ tested, the presentation of the results starts with this $Re$. Extensive study of the influence that the location of the external boundaries has on the solution for each $Re$ was performed. For convenience the numbers describing the location of the boundaries are given in the following order: upstream, downstream, lower, upper, and they represent multiples of the length of the plate. Although only the last steps of this procedure are presented here, it should be emphasized that the area ratio of the largest to the smallest numerical domain studied was around 12. Figure 5a gives the results of one of the first attempts to get the solution for $Re = 10000$. The (N) and (S) boundaries for both curves are located at $-1.0*L$ and $1.0*L$ respectively; only the location of the (W) is changed, located at $-.1*L$ - solid line - and at $-.2*L$ -square symbols, respectively. Fig. 5b shows how misleading the change of the location of only one boundary can be. In this figure, the solution obtained with the (W) boundary located at $-.5*L$ coincides with the solution obtained with the (W) boundary located at $-.4*L$ - square symbols. Although it seems that a grid-independent solution was obtained, Fig. 6 shows that this is not true. In this figure, it is obvious that both skin friction and pressure coefficients differ a lot when the location of the (N) and (S) boundaries are changed. The

3.0 Steady flow over 2D thick flat plates
location of the upstream boundary at 
-.5*L has been proven to be sufficient for a grid independent solution. The next figure (Fig. 7) clearly shows that a grid-independent solution was obtained. In this figure, the solid line corresponds to the domain with dimensions of the external boundaries: -0.8, 3.0, -4.0, 4.0 and the square symbols to the domain: -0.5, 3.0, -3.0, 3.0. The following numerical domain has been proven to give a grid-independent solution for Re 10000:

- (W) location - .5*L

- (E) location 3.0*L

- (N) location 3.0*L

- (S) location -3.0*L

Now that we are confident about the location of the boundaries of our domain, we can study the influence of the height of the first element. In Fig. 8, a comparison between the results obtained for the skin friction and pressure coefficient using two different heights of the first element near the leading edge of the plate are presented. The solid line corresponds to a height of the first element equal to .0005 of the length of the plate and, the square symbols to 3.0 Steady flow over 2D thick flat plates
a height equal to 0.0001 of the length of the plate. Although the ratio is 5:1, the results coincide. Even if the ratio is increased to 7:1, no change occurs in the results.

In Fig. 9, the skin friction coefficient for a plate with thickness, is compared to the one obtained for a plate without thickness. Clearly they agree, except in the regions close to the two ends. Comparing the skin friction coefficient for both cases, it is obvious that for the plate with no thickness the leading edge behaves like a singular point, causing a higher velocity field around it, which produces higher tangential stresses. As a result, the peak for this case is much higher than the one for the plate with thickness. The presence of the stagnation point on the lower surface of the plate, for the plate without thickness just after the leading edge also changes the pressure distribution near the leading edge (Fig. 10). Regarding the trailing edge, the geometry of the plate with thickness affects the tangential stresses and the pressure field. The stresses coincide for both cases over a large portion of the plate, however, as the geometric discontinuity near the trailing edge is approached, the stresses are locally increased and then decreased. The local acceleration of the fluid particles as they flow around the geometric discontinuity increases the stresses in that region. The flow is then decelerated over the entire length of the wedge due to the

3.0 Steady flow over 2D thick flat plates
adverse pressure gradient until it reaches the trailing edge, from which point a shear flow starts.

The existence of the stagnation point on the lower surface forces the fluid to increase its velocity in order to flow around the leading edge. The fluid, being unable to follow the sharp turn, separates and thus a recirculation bubble is established very close to the leading edge. In the case of the thin plate this situation is more severe, because the change of the geometry is more abrupt and the corresponding shear stresses are higher.

Fig. 11 gives a comparison between the solutions obtained for a 2D flat plate without thickness at 3.0 degrees and $Re = 10000$ and the Falkner-Skan [17] solution for the flow over a wedge of semiangle 3.0 degrees. This comparison is not meant to be exact, since the potential flow for the two cases is not the same. As expected, near the leading edge, the solution is completely different. The solution farther along the plate differs from that over the wedge by a relatively small amount due to the differences in the boundary conditions and the elliptic nature of the problem. At the trailing edge, the skin friction coefficient corresponding to the wedge solution is continuing to decrease, but the one corresponding to the solution of the finite flat plate is slightly increasing, since we have the singularity of the trailing edge.

3.0 Steady flow over 2D thick flat plates
Figure 12 shows the streamline pattern around the leading edge for the same Re. The height of the separation bubble is very small, and, for that reason, it cannot be seen on the top figure. The lower one presents a close view of the flow near the stagnation point and the increased magnitude of the velocities as the fluid is trying to flow around the leading edge.

The next case tested was the one corresponding to Re = 8000. Figure 13, presents the skin friction and pressure coefficients for two trial numerical domains. The solid line corresponds to the domain with dimensions: W=-.8*L, E=3.0*L, S=-3.5*L, N=3.5*L and the square symbols correspond to the domain with dimensions: -.65, 3.0, -3.0, 3.0 respectively. It is obvious that the pressure coefficient has already converged, and the same is true for the skin friction coefficient. However, the peak predicted for the stresses in the recirculation region is slightly different. Figure 14 shows the results for the same case, after changing the location of the (N) and (S) boundary. The two sets of results coincide for both the skin friction and pressure coefficients. So, the numerical domain which gives a grid-independent solution for Re = 8000 is:

- (W) location -0.8*L
- (E) location 3.0*L

3.0 Steady flow over 2D thick flat plates
o (S) location  -3.0xL

o (N) location  -3.0xL

Comparing this to the domain for Re = 10000, it becomes obvious that, as Re increases the upstream boundary can be located closer to the leading edge. Figure 15 shows the velocity profile for the upper and lower surface of the plate at two locations: 50 percent of the chord and trailing edge. The two profiles at the mid-chord are much different. The one corresponding to the lower surface seems to be closer to the velocity profile encountered in boundary layers. The thickness of the boundary layer is about \( \eta = 4.0 \), where the dimensionless distance \( \eta \) is defined as \( \frac{y}{2} \left( \frac{U_e}{\nu x} \right)^{0.5} \). The profile is more full than the one corresponding to the upper surface verifying the higher values observed for the skin friction coefficient of the lower surface. This difference of the velocity profile near the plate indicates also the difference in the pressure gradient, which being favorable along the lower surface, accelerates the flow and creates steeper velocity gradients. The velocity profile of the upper surface has a maximum at \( \eta = 3.6 \) exceeding the freestream velocity. After this maximum, the velocity approaches asymptotically the freestream velocity as the nondimensional distance normal to the plate increases. The behavior changes at the trailing edge. The maximum value of the ve-

3.0 Steady flow over 2D thick flat plates
Locity is nearly the same for both profiles. This value is closer to $u_e$ compared to the one corresponding to the mid-chord location. The flow is now fully developed, and the viscosity has smoothed the difference between $u_e$ and the maximum value. The velocity gradient normal to the plate is higher for the lower surface, so the tangent stress is also higher. The thickness of the boundary layer has also decreased.

Figure 16 contains the first results for $Re = 12000$. First of all, one must check the adequacy of the location of the external boundaries of the domain. The approximate rule that was used to relocate these boundaries with respect to the locations used for $Re = 10000$, is the square root rule - the square root of the ratio of the $Re$ times the location of the boundaries for the lower $Re$. Several other rules were tested with no success. In this figure, the solid line represents the solution obtained using the same numerical domain used for $Re = 10000$, while the square symbols represent the solution obtained when the boundaries of the domain are relocated according to the square root rule. Moving the external boundary in more than the amount mentioned above as in Fig. 17, slightly changes the solution for the recirculation area, thus proving that the solution for the recirculation is sensitive to the location of the upper and lower boundaries. Figure 18 compares the skin friction for two
combinations of heights of the first element. These combinations are: .0007 for the leading edge and 0.001 for the trailing edge, represented by the square symbols, and 0.0003 for the leading edge and 0.001 for the trailing edge represented by the solid line. Smaller values for the height of the first element are required for the leading edge region where the gradients are steep. The results are very satisfactory, and no further refinement is required. At the trailing edge the flow is fully developed, the height of the boundary layer has increased, and thus less nodes are required in the boundary layer. The prediction of the location of the reattachment point was not improved when a grid with higher density of nodes at the end of the recirculation bubble was used. The following numerical domain has been proven to give a grid-independent solution for Re = 12000:

- (W) location -.5*L
- (E) location 3.0*L
- (S) location 2.7*L
- (N) location 2.7*L

For all the cases presented until now, the convergence criterion used was 0.01. The numerical results did not
change when a lower convergence criterion was used. This is important for the results to be presented for higher Re.

Figure 19 presents results for Re = 15000. The new location of the external boundaries is again calculated using the square root rule. The results corresponding to the new location are represented by the solid line and coincide with the results obtained using the domain: -0.8, 3.0, -2.7, 2.7. The location of the upstream boundary was not changed for reasons concerning the construction of the grid; i.e. keeping the aspect ratio of the elements as low as possible. Actually, as we increase Re, the location of the upstream boundary can be moved closer to the leading edge. The results presented in Fig. 19 have been obtained using as convergence criterion 0.01, as before. The following figures (Figs. 20 and 21) show how the solution changes for a more strict criterion of convergence. The solution for most of the domain remains the same, but the solution for the separation bubble is slightly changed. The use of two different first elements, but with the same convergence criterion, does not change the solution. It should also be mentioned here that if we let the program iterate beyond the required accuracy, not only didn't we get lower residuals, but an oscillatory behavior of the residual occurs, with much bigger amplitude than the one observed for lower Re. This may be an indication that the nature of the flow is inherently unsteady, and the

3.0 Steady flow over 2D thick flat plates
steady solver has problems in converging to a steady solution as Re is increased.

Figures 22 to 24 correspond to Re = 18000, and should not mislead the reader. Here, the intention is to present the problematic behavior of the steady solver for this Re instead of presenting a "steady" solution. Figure 22 presents "solutions" obtained using four different heights for the first element with the solution stopped at a crude convergence criterion of 0.01. The complete failure of getting a grid-independent solution is obvious. Not only was the length of the recirculation region different, but also the shape of the curve corresponding to the skin friction coefficient changed. In Fig. 23, the same height of the first element was used but, the solution was stopped at two different convergence criteria and the corresponding "results" were plotted. It is obvious that the solution is dependent on the grid as well as the accuracy defined. Figure 24 may lead to the wrong conclusion, that by stopping the solution at a tight convergence criterion, a grid-independent solution can be obtained. This is so, because, this tolerance was satisfied only for a few iterations after which the residual increased again. All these observations lead to the conclusion that the steady state flow assumption is probably wrong for these conditions.

3.0 Steady flow over 2D thick flat plates
It is interesting to observe the behavior of the residuals in Fig. 25. for the different Re. It should be noted that the residuals corresponding to the different cases do not represent solutions started from a zero-velocity field, but from the previous lower Re, since this procedure gave a much faster convergence. So, while examining the residuals, note that this is the behavior of the solver when a solution vector near to the final solution is used to initiate the procedure. If the Stokes solution is used as an initial guess, the solver is still robust but a slower convergence is observed for the first steps.

The results for the integrated values of the skin friction and pressure coefficients along the plate for all the cases studied are compared in Fig. 26. Clearly, the integrated skin friction does not vary as the square root of the ratio of Re. This deviation from the classical boundary layer behavior can be explained better when the skin friction coefficients for the different Re are plotted on the same plot as in Fig. 27. As this figure indicates, the skin friction for the lower surface is decreasing much faster than the one corresponding to the upper surface. The flow on the upper surface is less constrained than the one on the lower surface; the presence of the solid wall forces the fluid particles to change direction faster thus producing steeper gradients. At the same time, the length of the recirculation

3.0 Steady flow over 2D thick flat plates
area increases, affecting directly the integrated value. Checking individually the integrated value for the upper and lower surface, one can see that although the integrated value for the lower surface follows the square root rule, the one corresponding to the upper surface deviates strongly. The situation is more complicated for the pressure coefficient.

3.4 Discussion of the results.

Some conclusions can be stated here for the steady flow over inclined flat plates at moderately high Re. Concerning the influence of the plate thickness, significant changes can be observed at the leading and trailing edges of the plate. This is natural, since, for the cases where the thickness is absent, these edges are only points. The sharp geometry creates higher velocities which make the gradients steeper. Although the solution is not changing much for most of the length of the plate, an accurate simulation of a real problem requires an exact description of the geometry at the edges. The thin plate solution can be accepted as a good approximation, if details of the flow near the edges are not required.

Next, the length of the recirculation area increases with the Re. Although the change seems to be close to linear, no accurate empirical rule can be extracted from Fig. 28.
The increase of Re seems also to have small effect on the pressure coefficient along the plate. The plots for Re between 8000 and 15000 for the pressure coefficient almost coincide. The location of the peak of the skin friction coefficient for all Re is very close to the point where the straight and the curved parts of the geometry of the nose of the plate are joined. It seems to be unaffected by Re. The existence of the maximum at that point can be attributed to the nature of the flow there. The fluid, coming from upstream, having just increased its velocity in order to flow around the leading edge, encounters the slowly moving fluid of the bubble accelerating it locally. This acceleration changes sharply the velocity field of the bubble. Since the height of the bubble is very small, the solid surface plays an important role in trying to decelerate the flow. This interaction between the high velocity external fluid and the solid surface produces high stresses in that region. At the trailing edge, the shape of the skin friction coefficient curve is highly determined by the geometry, and is not changing with Re, although the numerical value is changing.

The numerical solution for all the cases was performed completely in core on an IBM 3090 with vector facility. The memory required was between 45 and 55 megabytes. The CPU time required for the solution was of the order of 560 seconds for convergence tolerance of 0.0001. This number corresponds to

3.0 Steady flow over 2D thick flat plates
a solution using Stokes flow as a first guess. This procedure requires 2 successive substitutions and 4 Newton-Raphson or Quasi-Newton iterations for convergence.
4.0 UNSTEADY FLOW OVER 2D THICK FLAT PLATE

4.1 Boundary and initial conditions.

In Chapter 3, we saw the results obtained when a steady solution is assumed. In this chapter, the results from an unsteady simulation are presented to better understand the true nature of the problem especially at higher Re. Experiments have shown that for Re = 50000 and an angle of attack of 2.5 degrees, the flow is turbulent [18]. This observation along with the oscillatory behavior of the residuals for the "steady" higher Re cases that were examined, led to the examination of the steady solution for higher Re through an unsteady simulation.

The boundary conditions and meshes used here are the same as the ones used for the steady state solutions. As was already mentioned, any velocity field that satisfies the continuity condition can be used as initial condition, which means that any velocity field which is a steady state solution can be used. To obtain the current results, two kinds of unsteady simulations were performed. The first one was started with a velocity field corresponding to a steady state with Re lower than the final one. By increasing the freestream velocity, one reaches the final Re. The second
one was started using the steady solution at the final Re. No change of the freestream velocity is required in this case. Simply, marching in time, the changes— if any— occurring to the flow due to the unsteady terms present in the analysis are examined.

4.2 Results.

First, consider truly unsteady simulations starting from a different Re. Two such simulations were performed. The first one used as initial conditions the steady solution for Re=6600, and the second one the steady solution for Re = 8000. The inlet velocity was increased until Re = 10000 was reached. The increase of the inlet velocity was then stopped and the simulation continued for the fixed value of the inlet velocity corresponding to Re = 10000. In Fig. 29 a comparison between the steady solution for Re = 10000, and the final steady state for the unsteady one started from Re = 6600 is presented verifying that the flow is still steady for this final condition. Identical results were also obtained from the second simulation.

In Fig. 30, the vortex growth is presented at three different moments. Fig. 30a shows the flow field corresponding to the steady solution for Re = 6600. The flow tends to reverse, but no recirculation can be observed at this
point. As the freestream velocity increases in time, the flow is reversed just after the leading edge. This can be seen in Fig. 30b where the flow field corresponding to the unsteady solution for Re = 8200 is presented. As the freestream velocity is increased further, the corresponding Re is increased to 10000 in Fig. 30c and the magnitude of the velocities in the bubble increases considerably. Since the vortex is attached, a big time step is allowed.

Figure 31 compares the unsteady and steady solution for Re = 8000 expressed in terms of the skin friction and pressure coefficient. The unsteady solution is the one obtained at the moment that the increasing freestream velocity corresponds to Re = 8000. That implies that this solution does not represent the final steady state for this Re, since the freestream velocity continues to increase. Although a small difference can be observed in the recirculation region, the skin friction coefficient coincides for both cases, indicating that the velocity profile along the plate adjusts itself quickly to the changes of the inlet velocity. On the other hand, the pressure coefficient shows a lag in adjusting to the steady solution. In this simulation the inlet velocity corresponding to Re = 8000 was reached after ten seconds, which means that it would have taken this amount of time for the flow to reach Re = 8000 in a real life experiment. This allows for a smooth acceleration of the flow.

4.0 Unsteady flow over 2D thick flat plate
The next two figures (Fig. 32 and 33) present the unsteady calculations for two different moments for Re = 12000 starting from the steady solution for this Re. The amplitude of the oscillations observed in both skin friction and pressure coefficients seems to decrease as the simulation proceeds in time. This is an indication that the flow starts to become sensitive to the disturbances, but the nature of the flow is still steady, and thus these disturbances are finally damped.

The next unsteady simulation refers to the case of Re 18000, because this is the most probable case to give a truly unsteady solution based upon the earlier "steady" results. The simulation showed that up to t=21.52, the variable time-step algorithm was increasing the time step, reaching a maximum of 2.468. After that, the time step remained constant for a few steps, and then, beginning at t = 26.45 it started to decrease reaching a minimum of 0.0266 at t = 66.86. Obviously, the time step followed the physics of the problem. The decrease of the time step was due to the appearance of a small disturbance in the form of a vortex just before the point of reattachment. This initial instability of the flow near the end of the bubble is followed by a highly oscillatory motion of the fluid in this region due to the creation and destruction of a number of vortices. The point where the fluid reattaches is not stable anymore. Moreover,
the disturbances are transferred downstream due to the mean velocity of the fluid. As Figs. 34 and 35 show, the skin friction is changing between the different moments. This bursting motion of the fluid continues for a while as the curves in Fig. 34 indicate, but it seems that finally the flow tends to an unsteady periodic motion on the upper surface. This is indicated by the wave-like shape of the skin friction coefficient curve in Fig. 35. The same oscillatory motion of the velocity is observed in the wake of the plate, although the resolution of the grid away from the trailing edge is not adequate to accurately describe the small-scale motion of the fluid. Obviously the unsteady nature of the flow, especially near the end of the recirculation zone, makes the solution of the problem as a steady state one extremely difficult and explains the oscillations of the residual for the steady state solution. Note that the same grid was used for both steady and unsteady simulations.

4.3 Discussion

The presentation of the previous results indicates that the nature of the flow is changing as the Re increases. The hypothesis stated in Chapter 3 that the behavior of the steady solver is a direct result of the true nature of the flow has been established. The flow continues to be inher-
ently stable for at least Re < 12000. Recall that for this range of Re, during the steady state solution, very low convergence criteria were met with no problem. On the other hand, the unsteady simulation for Re = 18000 has shown a periodic, unsteady motion of the fluid on the upper surface of the plate. The origin of the instability is located in the bubble. As soon as the periodic behavior of the flow can be clearly identified, the time step changes only slightly from one step to the other. The results for the three unsteady simulations presented here do not allow for an identification of the exact value of the Re after which the flow becomes unsteady. Nevertheless, a value ~ 15000 seems to be a reasonable one.

The memory requirements for the unsteady simulations are the same as those for the steady ones, since the same grids are used. The CPU time varies from application to application. The number of QN iterations required for each time step depends on the nature of the flow. At moments where the nature of the flow is changing, the variable time step algorithm has to decrease the time step and this requires generally 3 to 4 iterations. If the changes of the flow field are not severe, then only one iteration per time step is required. The CPU time per iteration for a 321x35 grid is ~ 70 seconds.

4.0 Unsteady flow over 2D thick flat plate
5.0 RESULTS FOR STEADY FLOW OVER 3D FLAT PLATES.

5.1 Problem choice.

Most of the flowfields of practical interest are three dimensional. This reality makes it necessary for the engineer to be able to perform 3D calculations. Although numerical methods are available for such simulations, restrictions concerning the available memory and speed of the modern computers make, most of the time, these computations very expensive. The results presented here consider the flow over square flat, plates without thickness for low Re ( = 100 ) and two angles of attack, 3 and 8 degrees. The extreme case of aspect ratio = 1 was chosen since it introduces strong three dimensional effects.

Some of the reasons for which such a low value for Re was chosen are the following:

- We wanted to study problems with strong viscous effects.
- In the limit, information concerning viscous flows of high Re can be obtained using inviscid methods such as the vortex lattice method.
Flows characterized by low Re can be resolved with coarser grid since the gradients are not very steep.

The values of 3. and 8. degrees were chosen, since the behavior of this idealized "wing" with low angles of attack avoiding massive stall was of interest.

5.2 Grid choice.

The numerical domain is a parallelepiped, and due to the left/right symmetry, only half of the flow field is simulated. The grid is more dense near the edges of the plate and normal to the plate in order to be able to capture the steep gradients in these areas. The nodes on either side of the plate are co-located but they assume different values. The boundaries are located as follows:

- **(W) location 1.*L**
- **(E) location 3.*L**
- **(N) location 1.*L**
- **(S) location -1.*L**
- **(R) location 1.5*L (from the centerplane)**

5.0 Results for steady flow over 3D flat plates.
The grid used for both cases was 51*26*19 (length*height*lateral direction) as shown in Fig. 36. The total number of nodes was 24475 and the number of elements was 21600.

The solution could not be checked for grid-independence, since the number of nodes required for such a test was out of the question. The grid used was the biggest possible for the available facilities. The dimensions of the grid and the distribution of the nodes were based on:

a) previous experience [16] obtained from the solution of 3D flow over a flat plate at zero angle of attack

b) tests with two-dimensional grids

c) comparisons of the pressure field results with those obtained using the vortex lattice method.

Results obtained from the tests concerning the 2D cases are presented in Fig. 37 in terms of skin friction and pressure coefficients. In this figure, the grid and domain independent solution is compared to the one obtained when the centerplane of the 3D grid is used as a two dimensional mesh. The latter solution is obtained using linear interpolation for the velocity - the same order of interpolation is used for the 3D flow also - which implies constant pressure in each element. The comparison for the skin friction coefficient is very good. On the other hand, the comparison for the pressure coefficient is not as good. Thus, one is led

5.0 Results for steady flow over 3D flat plates.
to suspect that the solution for the pressure in the 3D flow is of questionable accuracy. The difference observed in the two dimensional solution is not due to the difference in the interpolation for the velocities and pressure, but, due to the different dimensions of the numerical domain. If the domain -2.5,3.0,-5.0,5.0 is used, both linear and quadratic interpolation gives identical result for the pressure and skin friction coefficients leading to the conclusion that the location of the external boundaries of the numerical domain is the most important factor for a grid-independent solution.

To study this problem further, the pressure difference along the centerline of the 3D plate, obtained from the FEM solution was compared to the one obtained from the Vortex Lattice Method (VLM) [20] in Fig. 38. The comparison is very good for both angles of attack. The differences observed in the vicinity of the leading edge should be attributed to the coarse discretization of the VLM. As the discretization becomes finer, the vortex lattice solution approaches the one obtained from the FEM. The VLM method is essentially domain free. One should assume that if the 3D domain is giving good results, that is because it is less restrictive on the flow than the 2D case. This must be a result of the three dimensional relief of the flow in the case of low aspect ratio. In other words, the boundaries are numerically "far enough" from the plate. Since traction-free

5.0 Results for steady flow over 3D flat plates.
conditions are imposed on the external boundaries, if the latter assumption is correct, the pressure there should be nearly the freestream one. Checking the pressure values on these boundaries, it was found that this was true, and thus the 3D domain is not as restrictive as the 2D domain study implies.

5.3 Boundary conditions.

The boundary conditions were:

- upstream boundary (W) \( U_\infty = 1.0, \ V_\infty = 0.0, \ W_\infty = 0.0 \)
- "Traction-free" conditions on the downstream (E), upper (N), lower (S), and outer boundary (R), i.e., \( t_i = \sigma_{ij} n_j = 0 \).
- centerplane (CP) \( w = 0 \) (symmetry of the flow field), \( t_x = t_y = 0 \).

5.4 Solution strategy.

The strategy for obtaining the solution consisted of two SS and two to three NR iterations. A zero-velocity field was used as the first guess for the iterative scheme. The solution required a disk space of 1.6 Gigabytes and the CPU time.
per iteration was between 12500 seconds (SS) and 16000 seconds (NR), on an IBM 3090 with vector facility. A maximum number of 5 iterations was required for achieving a convergence tolerance of 0.01.

5.5 Results.

Figure 39 presents the results for the velocity field at two locations near the trailing edge of the plate, i.e., x=0.95 and x=1.05, for the case a = 3.0 degrees. The figure presents the projection of the velocity vector on a plane normal to the plate at the two locations, seen from the wake of the plate. In reality, the components of the velocity on this plane are small, and for that reason, the velocity vectors are magnified in this figure. The existence of the vortex attached at the tip of the plate is obvious. The influence of the vortex on the flow below the plate is much stronger than on the flow above plate; this is indicated by the change of direction of the velocity vectors. The velocity vectors, located at a distance greater than half the length of the plate above the plate are parallel to the centerplane, unlike the ones below the plate which have a relatively large z-component even near the lower boundary of the region. The influence of the vortex on the velocity field can also be seen in Fig. 40 where the velocity fields

5.0 Results for steady flow over 3D flat plates.
at \( x = 1.05 \) are compared for the two angles of attack. For \( a = 8.0 \) degrees, the influence of the vortex is bigger, i.e. the vectors are becoming parallel to the centerplane after one length of the plate. This is expected, since the higher inclination of the plate is producing a stronger vortex at the side edge of the plate. In Fig. 41, a strong outward component of the velocity is observed for a plane parallel to the plate and located 0.1 lengths below the plate. In the same figure, the presence of a lateral boundary layer along the edge of the plate is observed.

The next two figures (Fig. 42 and 43) compare the pressure field just above and below the plate for \( a = 8^\circ \), and the pressure field just below the plate for the two angles of attack, respectively. The presence of the vortex at the side edge of the plate changes considerably the isobars on the two surfaces of the plate. The angle of attack changes the magnitude of the isobars, but seems to have small influence on the general qualitative picture. Compare also the high concentration of pressure lines at the leading and trailing edge of the plate with the ones at the side edge. This indicates the presence of steeper pressure gradients at the first two edges compared to the latter. The presence of these gradients, especially the one at the leading edge, forces the fluid to try to avoid the plate and flow in the sidewise direction.

5.0 Results for steady flow over 3D flat plates.
Next, some information about the vorticity field is presented. Figure 44 compares the iso-vorticity curves just above the plate for the two angles of attack. The three edges of the plate are locations where a large amount of vorticity is created, as the concentration of the curves of equal vorticity indicates. The similarity of the vorticity field on the plate is interesting in that the curves for both angles of attack follow closely the shape of the plate. The downstream advection, more obvious in the region of the tip vortex, shows up only for a few contour lines, since the rest are diffused by the viscosity. In fig. 45, the vorticity is plotted on two planes, the centerplane and one parallel to that but, located at the side edge of the plate. The differences are obvious; for the centerplane, the vorticity is mainly produced at the leading and trailing edge, unlike the case of the side edge which is a continuous source of vorticity.

Figure 46 compares the skin friction coefficient for the 2D and 3D cases. These are two sets of 2D results. One is for grid independent solution and the other for the solution obtained when the centerplane of the 3D grid is used as a two dimensional grid. The skin friction for the upper surface seems to be only slightly affected, indicating a similarity, at least near the wall, of the velocity profiles on the upper surface of the plate along the centerline. The

5.0 Results for steady flow over 3D flat plates.
skin friction on the lower surface is different. Higher values are observed for the two dimensional flow. This result contradicts the result observed for the skin friction coefficient for a three dimensional flow over a flat plate at no angle of attack, where the skin friction was higher than the one observed for the two dimensional flow [16]. That result was attributed to the formation of a thinner boundary layer due to the divergence of the streamlines. For an inclined plate, comparison between the two flows concerning the thickness of the boundary layer reveals that the change is small. The profiles are almost identical for the upper surface. For the lower surface, the velocity profile corresponding to the 3D flow is slightly shifted upwards explaining the lower values of the skin friction coefficient.

On the other hand, the pressure coefficient is much different as shown in Fig. 47. The area between the curves corresponding to the two surfaces, and their relative position is different. Comparing the area under the curves, it is obvious that the lift is higher for the 2D case compared to the centerplane for 3D. It should be noticed here that the lift obtained from the two dimensional simulation is less than $2\pi\alpha$, the result predicted by the thin airfoil theory. The same figure indicates that the pressure coefficient along the side edge of the plate, shown as a dashed line, increases nearly linearly in the streamwise direction between 10 and 5.0 Results for steady flow over 3D flat plates.
85 percent of the length of the plate. The next figure (Fig. 48), shows a comparison for the lift produced at two locations in the lateral direction on the plate expressed in terms of the pressure jump between the upper and lower surface of the plate, for both angles of attack. Clearly the lift at 8.0 degrees angle of attack is much higher than that at 3.0 degrees angle of attack. For both cases, the local difference of the pressure coefficient corresponding to the centerplane is compared to that corresponding to the plane located at the midspan. As the flow becomes fully developed along the plate, these differences diminish, and the local values of the pressure difference at both locations become the same. The figure also verifies that the pressure difference approaches zero at the leading and trailing edge. The numerical values are not exactly zero since the nodes corresponding to the edges are not shown here. Figure 49a presents the variation of the total lift coefficient versus the angle of attack. Although Re is very low, the lift is increasing linearly with the angle of attack. The effect of the aspect ratio on the lift predicted for this idealized "wing" does not agree with the inviscid lifting line theory results, for the cases studied here. The very low aspect ratio used and the strong viscous effects are two reasons for this behavior. Figure 49b presents the variation of the local lift coefficient along the spanwise direction. The co-

5.0 Results for steady flow over 3D flat plates.
efficient decreases smoothly until the side edge of the plate is reached. Since the pressure difference at the edge is zero, a sharp drop of the lift is observed in the vicinity of the side edge.

Figure 50 compares the skin friction for the upper and lower surface of the plate along the centerline. Obviously, the difference gets bigger as the angle of attack increases. As the angle of attack increases, the skin friction coefficient for the lower surface increases, since the flow is accelerated more. Also, the skin friction coefficient for the upper surface is decreasing, since the adverse pressure is increasing, causing a higher deceleration of the flow.

Figures 51 and 52 present the change of the velocity profiles along the side edge and the trailing edge of the plate. Consider first the w-component. As can be seen in Fig. 51, for the side edge for x=0.3333 a big positive w-component is generated only for the lower surface. The w-component for the upper surface, at the same location, is much smaller and negative. This is due to the inclination of the plate. The fluid near the lower surface, being more restricted due to the presence of the rigid boundary, reacts to this restriction by increasing the only unrestricted velocity component (w). Approaching the trailing edge, the velocity profile tends to become more symmetric, since the vortex has relaxed the differences between the two surfaces.

5.0 Results for steady flow over 3D flat plates.
Next, look at the $u$ velocity profiles along the trailing edge. The profiles are changing only near the side edge of the plate. The comparison of the profiles for the locations 0.0 and 0.25 shows almost identical results. The last profile, at the location $z=0.5$ (the plate edge), is the only one that differs considerably from the rest. It is also the only one of the three that lies fully in the vortex.

Figure 53 presents the growth and the location of the vortex in the wake for angle of attack $= 8.0$ degrees. The first figure presents the projection of the velocity on a plane normal to the plate, while the second and the third are on a plane normal to the free stream velocity. The change of the inclination of the planes was done on purpose, since the core of the vortex tends to align itself with the free-stream velocity as the distance from the trailing edge increases. The lines observed on the left of the second and third figure are the outline of the plate as seen from the rear. The three figures present the flow field as it can be seen from an observer located in the wake. The diameter of the vortex is increasing as the vorticity is diffused in the wake. At the same time, a combination of a general downward motion along with an outward motion of the whole system is observed. A strong downward flow-field is established one length of the plate away from the trailing edge. This is the "down wash" of finite wing theory fame.

5.0 Results for steady flow over 3D flat plates.
CONCLUSIONS

The steady and unsteady flow around 2D and steady flow around finite 3D plates at angle of attack over a range of Reynolds numbers have been studied in this thesis. First the main 2D and then the 3D results are summarized below.

2D cases:

The nature of the flow for an angle of attack = 3.0 degrees was found to be inherently steady for Re < 12000. The skin friction and pressure coefficients obtained from both a steady and an unsteady simulation, for this range of Reynolds number, were identical. Convergence difficulties of the steady solver, appearing in the form of oscillating residuals, for Re = 15000 and 18000 were proved to be a result of the unsteady nature of the flow.

The length of the recirculation on the top surface near the leading edge increases with the Re but no empirical law was extracted from the results for the reattachment point.

The behavior of the quasi-steady pressure coefficient for Re = 8000 indicated a lag of the pressure in adjusting to the steady state solution.

The ratio of the skin friction coefficients for two different Re is not proportional to the -0.5 power, as one may expect from classical boundary layer theory. The dependence
is not clear from the results. The existence of the recirculation zone is an important factor which influences this result for both thin and thick plates.

3D cases:

Calculations for three dimensional flows are still very expensive. The introduction of the w-component of the velocity as an extra unknown at each node along with the introduction of nodes in the z-direction increases dramatically the number of equations to be solved, thus increasing both the memory and CPU time required for those simulations.

The solution is strongly influenced by the existence of the tip vortices. Both the velocity field and the pressure field are altered significantly over the plate and in the wake.

The lift produced in the 3D case is lower than the one produced in the 2D case, but it increases linearly with the angle of attack for the angles examined here. The prediction for the lift does not agree with results obtained from the lifting line theory.

Values of the pressure difference between the upper and lower surface along the centerplane predicted from FEM are in very good agreement with those predicted from the vortex lattice method.

The skin friction coefficient for the lower surface predicted from the three dimensional solution is lower than the
corresponding one for the two dimensional flow. This result contradicts previous observations of higher values for the skin friction coefficient on the centerplane in a 3D flow over a flat plate at zero incidence [16].

The skin friction coefficient for the lower surface increases along the centerline of the plate with the angle of attack, while the one for the upper surface decreases. This is a result of the acceleration of the flow on the lower surface - the pressure gradient becomes more favorable - and of the deceleration of the flow on the upper surface - the pressure gradient there becomes more adverse.

The velocity profiles along the trailing edge are altered considerably only near the side edge of the plate due to the presence of the tip vortex.

The diameter of the tip vortices increases in the wake. A strong downward flow-field induced by the vortices is established in the wake of the plate.

More work should be devoted to obtaining: 1) unsteady solutions for higher Re than these presented here because the results can be very helpful, if someone wants to check how the numerical solution behaves as we approach the range of Re where transition of the flow from laminar to turbulent occurs. 2) steady solutions for higher Re for the 3D cases. 3) unsteady solutions for 3D cases.
Figure 1. Quadrilateral and brick elements.
Figure 2. "C-type" grid: Used for thick plates.
Figure 3. Geometry of the thick plate.
Figure 4. "H-type" grid. Used for thin plates.
Figure 5. Skin friction coefficient for $Re = 10000$, $a = 3.0$ degrees. Solutions obtained varying the location of the upstream boundary ($W$). Thick plate.
Figure 6. Skin friction and pressure coefficients for $Re = 10000$, $a = 3.0$ deg.: Grid-dependent solutions for two different locations of external boundaries. Thick plate.
Figure 7. Domain independent solution for $Re = 10000$, $a = 3.0$ deg.: Skin friction for: a) the entire plate b) The recirculation region.
Figure 8. Skin friction and pressure coefficients for $\text{Re} = 10000$, $\alpha = 3.0$ deg.: Solutions for two different first element heights.
SKIN FRICTION COEFFICIENT
MESH: -.8*L,3.0*L -3.0*L,3.0*L

Figure 9. Skin friction coefficient for thick and thin plates: $Re = 10000$, $a = 3.0$ degrees.
Figure 10. Pressure coefficient for thick and thin plates: Re = 10000, α = 3.0 degrees.
Figure 11. \( Re = 10000, \alpha = 3.0 \) degrees.: comparison between the solution for a 2D thin flat plate and a wedge of semi-angle of 3.0 degrees.
Figure 12. Streamline pattern around the leading edge: $Re = 10000$, $a = 3.0$ degrees.
Figure 13. Skin friction and pressure coefficients for thick plates: Two choices of computational region. Re = 8000, a = 3 deg.
Figure 14. Grid independent solution for $Re=8000$, $\alpha=3.0$ degrees
Figure 15. Velocity profiles at the midchord and trailing edge. Thick plate. Re = 8000, $a = 3.0$ degrees.
Figure 16. Skin friction and pressure coefficient for $Re = 12000$, $a = 3.0$ degrees. Thick plate. Domain independent solution.
Figure 17. Re = 12000, a = 3.0 degrees. Effect of the location of the external boundaries on the solution of a thick plate.
Figure 18. Skin friction coefficient for different first element height. Thick plate. Re = 12000, α = 3.0 degrees.
Figure 19. Skin friction and pressure coefficients for Re = 15000, a = 3.0 deg.: Effect of the dimensions of the numerical domain.
Figure 28. Skin friction coefficient for Re = 15000, a = 3.0 degrees. Study of the influence of the user-prescribed convergence tolerance on the solution.
Figure 21. \( Re = 15000 \), \( \alpha = 3.0 \) degrees, Skin friction coefficient: Effect of the first element height on the solution. Convergence tolerance < \( 5 \times 10^{-5} \).
Figure 22. Apparent effect of the height of the first element on the solution. Solution stopped at a crude convergence tolerance. The solution is not converged.
Figure 23. Comparison between two solutions for $Re = 18000$. Stopping at two different convergence criteria. This solution was not finally converged.
Figure 24. Apparent effect of the height of first element. Solution stopped at tight convergence tolerance. The solution is not converged.
Figure 25. Behavior of the residuals corresponding to the solution of different Re numbers.
Figure 26. Skin friction and pressure coefficients vs. Re.: Integrated values for thick plates.
Figure 27. Local values of skin friction coefficient along the plate vs. Re.: Thick plates.
Figure 28. Recirculation bubble length vs. Re. Thick plates.
Figure 29. Steady state for the unsteady simulation starting at Re = 6600 and ending at Re = 10000. Compared to steady simulation at Re = 10000. Thick plates.
Figure 30. Vortex formation and growth at the leading edge. Thick plate: a) Re = 6600, b) Re = 8200, c) Re = 10000
Figure 31. Steady vs. unsteady simulation for Re = 8000, t = 10. Initial condition: Steady flow for Re = 6600.
Figure 32. Steady vs. unsteady simulation for Re = 12000, t = 3.35. Initial condition: steady flow for Re = 12000.
Figure 33. Steady vs. unsteady simulation for $\text{Re} = 12000$, $t = 5.39$. 

FIGURES
Figure 34. Skin friction coefficient vs. x for Re = 18000. Part of the recirculation region plotted. Unsteady simulation.
Figure 35. Skin friction coefficient for the region after the recirculation zone: \( \text{Re} = 18000, \alpha = 3.0 \) degrees. Unsteady simulation.
Figure 36. 3D mesh.
Figure 37. Skin friction and pressure coefficient for 2D flow: Influence of the grid and the domain.
Figure 38. Pressure coefficient for the centerplane: FEM vs. Vortex Lattice Method (from Ref. [20]).
Figure 39. Vertex at the side edge of the plate. $Re = 100$, $a = 3.0$ degrees. Flow field observed from the wake.
Figure 40. Vortex formation for the two angles of attack.
Figure 41. Top view of the flow field at the side edge of the plate.
Figure 42. Pressure field. Upper and lower surfaces of the plate. $Re = 100$, $a = 8.0$ degrees.
Figure 43. Pressure field. Lower surface for $a = 3.0$ and $8.0$ degrees.
Figure 44. Vorticity field $0.0001*L$ above the plate: Two angles of attack, 3. and 8. degrees.
Figure 45. Vorticity field, centerplane and plane at the side edge.
SKIN FRICTION COEFFICIENT, RE = 100, θ = 3. DEG
(CENTERPLANE)

![Graph showing skin friction coefficient vs. x-location]

3D SOLUTION, RE = 100, A = 3. DEG

2D SOLUTION, QUADRATIC ELEM., D: -2.5, 3.0, -5.5.

2D SOLUTION, LINEAR ELEM., D: -1.3, -1.1.

Figure 46. Skin friction. 2D versus 3D calculations (centerplane).
Figure 47. Pressure coefficient. 2D vs 3D calculations (centerplane).
Figure 48. Lift production, expressed in terms of pressure difference.
Figure 49. Re = 100, $a = 3.0$ and $8.0$ degrees. 

a) Lift coefficient versus $a$ (degrees). Re = 100.
b) Variation of pressure difference along the span of the plate.
Figure 50. Skin friction coefficient on the centerplane, 3D flow. $a = 3.0$ and 8.0 degrees.
VELOCITY PROJECTION ON THE X–Y PLANE
FOR THREE POSITIONS ALONG THE TRAILING
EDGE OF THE PLATE
3D, RE=100, ANGLE OF ATTACK =8.0 DEGREES

Figure 51. W-component of the velocity along the side edge.
Figure 52. Velocity profiles along the trailing edge.
Figure 53. $Re = 100$, $\alpha = 8.0$ degrees: Growth of the tip vortex for three locations in the wake.
REFERENCES


VITA

Evangelos Hytopoulos was born in June 1965 in Athens, Greece. He studied Naval Architecture and Marine Engineering at the Technical University of Athens and obtained his Diploma in 1988. The same year, he enrolled at VPI & SU in the Aerospace & Ocean Engineering department. He pursued his graduate work under the direction of Dr. Joseph A. Schetz. Mr. Hytopoulos is a member of the Society of Naval Architects and Marine Engineers (SNAME) and the American Institute for Aeronautics and Astronautics (AIAA).