Master Thesis

Development of Body and Viscous Contribution to a Panel Program for Potential Flow Computation

Aero Dynamic Analysis and Preliminary Design Tool

Erik Nordin

March 2006

LiTH-IKP-EX--062357--SE

Linköpings universitet
INSTITUTE OF TECHNOLOGY

Division of Fluid and Mechanical Engineering Systems
Department of Mechanical Engineering
Linköping University, SE-581 83 Linköping
ACKNOWLEDGEMENTS

This thesis is part of my master degree in Mechanical engineering at Linköping Institute of Technology, Sweden, where I have majored in Aircraft Technology. The thesis has been carried out at the department of Aerodynamics and Flight mechanics at Saab Aerosystems in Linköping.

During my work at Saab I have learned how the development and design process of airplanes is performed. I would like to thank Saab Aerosystems for the opportunity to do this thesis. I would also like to give my greatest gratitude’s to my advisors at Saab Aerosystems, Roger Larsson and Per Weinerfelt. Roger Larsson with his multifaceted knowledge in aerodynamics and experience from earlier master thesis at Saab, for help and support during the work. Per Weinerfelt, that with his expertise in computational aerodynamics answered many of my questions and helped me in finding and understanding different calculation methods and for help with the programming in Matlab.

I would like to give my thanks to my examiner Petter Krus, advisor Christopher Jouannet at Linköping Institute of Technology for good input to the report.

Linköping 2006-02-20

Erik Nordin
ABSTRACT

The aim of this thesis was to develop a potential flow calculation model which includes computation of flow around aircraft bodies (fuselage, engines) and a boundary layer method which calculates the viscous effects over the aircraft wings. The models developed will be merged with of an already existing panel program developed at Saab, Linköping, Sweden. Different methods have been studied but the basis of the work have been to develop a model using a panel method which can provide results from a simple geometry description, with short calculation time and hence be used in early design phases. In this thesis Matlab have been used as programming language, this to ensure that future development and maintenance is possible.

The body model uses a panel method where the flow domain is divided into an inner and an outer part where the outer problem uses a three dimensional panel description while the inner problem performs two dimensional calculations. The inner and outer problems are separated by an arbitrarily shaped reference box. The inner area is divided into a number of cross sections which are described by line segments. With the help of these the two dimensional cross flow is obtained. This result is connected to the outer part trough boundary conditions and the entire three dimensional flow domain can be determined.

The resulting body program is limited to aircraft bodies with a slenderness ratio less than 1/5. Higher values violate the model assumption. The number of cross sections needed to describe a body of one unit length is between 80-150 and the number of line segments needed for one cross sections is 20 for the inner boundary and 40 line segments for the outer. This configuration gives results with acceptable accuracy within a computation time less than 15 seconds/body.

The viscous effects around the aircraft wings are modelled with a two dimensional boundary layer model where the boundary layer displacement thickness over the wing profile is calculated with two different methods depending on if the flow in the boundary layer is laminar or turbulent. The computed displacement thickness is then added to the wing profile geometry and new pressure distributions are computed on the modified geometry.

The computed pressure distributions including the viscous effects show better agreement with results from experimental wind tunnel tests than the inviscid without boundary layer contribution. Separation is not modelled and neither the large effects this has on the pressure distribution. The model gives useable results up to 15-20 degrees angle of attack; at higher angles the separated regions are so large that the model is not valid anyway.

The work was performed at the department of applied aerodynamics and flight mechanics at Saab Aerosystems during the period 2005-08-24 – 2006-01-31.
SAMMANFATTNING


De viskösa effekterna kring flygplanets vingar är modellerade med en tvådimensionell gränsskiktetsmodell där gränsskiktets förträngningstjocklek över vingprofilen beräknas med två olika metoder beroende på om strömningen i gränsskiktet är laminär eller turbulent. Den beräknade förträngningstjockleken adderas sedan till vingprofilens geometri och nya tryckfördelningar beräknas på den modifierade geometrin.

De beräknade tryckfördelningarna innehållande de viskösa effekterna visar bättre överensstämmelse med resultat från experimentella vindtunnelprov än den inviskösa utan gränsskiktssbidrag. Separation modelleras inte och därför inte heller de stora effekterna de har på tryckfördelningen. Modellen ger användbara resultat upp till 15-20 graders anfallsvinkel, vid högre är de separerade områdena så stora att modellen ej är giltig.

# TABLE OF CONTENTS

1 **INTRODUCTION** .............................................................................................................................................. 1

1.1 **BACKGROUND** .................................................................................................................................................. 1

1.2 **OBJECTIVES** ....................................................................................................................................................... 1

1.3 **LIMITATIONS** ....................................................................................................................................................... 2

1.4 **READING INSTRUCTIONS** ................................................................................................................................. 2

2 **BODY MODEL** .................................................................................................................................................... 3

2.1 **PROBLEM DESCRIPTION AND REQUIREMENTS OF BODY CONTRIBUTION** .................................................. 3

2.2 **THEORETICAL BACKGROUND** ............................................................................................................................ 4

2.2.1 **General Flow Theory** ...................................................................................................................................... 4

2.2.2 **Potential Theory** .............................................................................................................................................. 5

2.2.3 **Elementary Flow Cases** .................................................................................................................................... 6

2.2.4 **Panel Methods** .................................................................................................................................................. 7

2.3 **METHOD** ............................................................................................................................................................. 8

2.3.1 **The Inner Problem** ........................................................................................................................................... 8

2.3.2 **Calculation of the Velocity at the Body Surface** ............................................................................................... 11

2.3.3 **Velocity Contribution due to Vortices** ............................................................................................................. 12

2.3.4 **Calculation of Boundary Condition at the Body Surface** .................................................................................. 12

2.3.5 **The Outer Problem** ........................................................................................................................................... 12

2.3.6 **Calculation of the Pressure Distribution Cp** .................................................................................................... 13

2.4 **DATA MANAGEMENT** ........................................................................................................................................ 14

2.4.1 **Program Structure** ......................................................................................................................................... 14

2.4.2 **Geometry Treatment** ....................................................................................................................................... 15

2.5 **SUMMARY OF COMPUTATION RESULTS USING THE BODY MODEL** ........................................................... 17

2.5.1 **Discussion of Body Model** ............................................................................................................................. 18

3 **BOUNDARY LAYER MODEL** ............................................................................................................................... 19

3.1 **THEORETICAL BACKGROUND** ......................................................................................................................... 19

3.1.1 **Boundary Layer Theory** .................................................................................................................................. 19

3.1.2 **Boundary Layer Around Airfoils** ................................................................................................................... 19

3.1.3 **The Boundary Layer Equations** ...................................................................................................................... 20

3.1.4 **Boundary Conditions, Boundary Layer Thickness** ........................................................................................ 21

3.2 **LAMINAR SOLUTION METHODS** ........................................................................................................................ 24

3.2.1 **Thwaites’ Method** ............................................................................................................................................. 25

3.2.2 **Von Karman and Pohlhausen’s Method** .......................................................................................................... 27

3.3 **TURBULENT SOLUTION METHOD** ..................................................................................................................... 29

3.3.1 **Background** ....................................................................................................................................................... 29

3.3.2 **Head’s Method for Turbulent Flow** .................................................................................................................. 29

3.3.3 **Prediction of Transition and Separation** .......................................................................................................... 31

3.4 **SOLUTION PROCEDURE OF THE BOUNDARY LAYER PROBLEM** .................................................................. 31

3.5 **DATA MANAGEMENT** ........................................................................................................................................ 32

3.5.1 **Program Structure** ......................................................................................................................................... 32

3.5.2 **Method Modifications** .................................................................................................................................... 32

3.6 **SUMMARY OF COMPUTATION RESULTS USING THE BOUNDARY LAYER MODEL** .................................... 33

3.7 **DISCUSSION OF BOUNDARY LAYER MODEL** .................................................................................................. 33

4 **CONCLUSIONS** ..................................................................................................................................................... 35

5 **FUTURE WORK** .................................................................................................................................................... 37

6 **REFERENCES** ...................................................................................................................................................... 39

7 **APPENDICES** ......................................................................................................................................................... 41

APPENDIX A: **VALIDATION OF BODY CONTRIBUTION** ............................................................................................ 41

Validation 1: 2D Flow Over a Wing Body Configuration ......................................................................................... 41

Validation 2: 3D Axial Flow Over a Parabolic and an Elliptic Shaped Body ............................................................... 46

APPENDIX B: **VALIDATION OF BOUNDARY LAYER MODEL** ..................................................................................... 50

Validation 1: Flow Over a Flat Plate with Zero Incidence .......................................................................................... 50
# TABLE OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Elementary flows</td>
<td>6</td>
</tr>
<tr>
<td>2</td>
<td>The panel method applied on an entire aircraft</td>
<td>7</td>
</tr>
<tr>
<td>3</td>
<td>Example of an inner problem</td>
<td>8</td>
</tr>
<tr>
<td>4</td>
<td>Inner boundary with panels</td>
<td>10</td>
</tr>
<tr>
<td>5</td>
<td>Body section with T-tail configuration</td>
<td>16</td>
</tr>
<tr>
<td>6</td>
<td>Section containing more than one element</td>
<td>16</td>
</tr>
<tr>
<td>7</td>
<td>Transition from laminar to turbulent boundary layer</td>
<td>20</td>
</tr>
<tr>
<td>8</td>
<td>Laminar separation</td>
<td>20</td>
</tr>
<tr>
<td>9</td>
<td>Displacement thickness</td>
<td>22</td>
</tr>
<tr>
<td>10</td>
<td>Boundary layer section used for deriving the momentum integral in two dimensional flow</td>
<td>23</td>
</tr>
<tr>
<td>11</td>
<td>Concept of Entrainment velocity</td>
<td>29</td>
</tr>
<tr>
<td>12</td>
<td>Wing - Slender body combination</td>
<td>41</td>
</tr>
<tr>
<td>13</td>
<td>Contour graph - Theoretical solution</td>
<td>43</td>
</tr>
<tr>
<td>14</td>
<td>Contour graph - Panel method</td>
<td>44</td>
</tr>
<tr>
<td>15</td>
<td>Contour graph - Wing Body - Theoretical solution</td>
<td>45</td>
</tr>
<tr>
<td>16</td>
<td>Contour graph - Wing Body - Panel method</td>
<td>45</td>
</tr>
<tr>
<td>17</td>
<td>Cp along parabolic body of revolution</td>
<td>47</td>
</tr>
<tr>
<td>18</td>
<td>Cp along elliptic body of revolution</td>
<td>48</td>
</tr>
<tr>
<td>19</td>
<td>BL thickness for a flat plate at laminar conditions</td>
<td>51</td>
</tr>
<tr>
<td>20</td>
<td>Displacement thickness for a flat plate with laminar BL</td>
<td>52</td>
</tr>
<tr>
<td>21</td>
<td>BL thickness for a flat plate at turbulent conditions, Reynolds numbers less than $10^7$</td>
<td>52</td>
</tr>
<tr>
<td>22</td>
<td>Displacement thickness for a flat plate with turbulent BL</td>
<td>53</td>
</tr>
<tr>
<td>23</td>
<td>BL thickness for a flat plate at turbulent conditions, Reynolds numbers higher than $10^7$</td>
<td>53</td>
</tr>
<tr>
<td>24</td>
<td>Geometry of NACA LS(1)-0417 mod., boundary layer at 8 degree angle of attack</td>
<td>54</td>
</tr>
<tr>
<td>25</td>
<td>Cp distribution around NACA LS(1)-0417 mod. airfoil at 8 degree angle of attack</td>
<td>55</td>
</tr>
<tr>
<td>26</td>
<td>Cp distribution around NACA 0417 mod., 2 iterations</td>
<td>55</td>
</tr>
<tr>
<td>27</td>
<td>Example aircraft 1, civil passenger aircraft</td>
<td>60</td>
</tr>
<tr>
<td>28</td>
<td>Example aircraft 1, civil passenger aircraft with outer reference box</td>
<td>60</td>
</tr>
<tr>
<td>29</td>
<td>Example aircraft 2, two engine aircraft with V-tail and reference boxes</td>
<td>61</td>
</tr>
<tr>
<td>30</td>
<td>Example aircraft 3, UAV with reference box</td>
<td>61</td>
</tr>
<tr>
<td>31</td>
<td>Boundary layer around NACA 4412 at 8 degree angle of attack</td>
<td>62</td>
</tr>
<tr>
<td>32</td>
<td>Boundary layer around NACA 0010 at 5 degree angle of attack</td>
<td>62</td>
</tr>
<tr>
<td>33</td>
<td>Displacement thickness - Upper surface NACA 0010, alpha=5 degrees</td>
<td>63</td>
</tr>
<tr>
<td>34</td>
<td>Displacement thickness - Lower surface NACA 0010, alpha=5 degrees</td>
<td>63</td>
</tr>
</tbody>
</table>
NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>u, V</td>
<td>Flow velocity</td>
<td>[m/s]</td>
</tr>
<tr>
<td>c</td>
<td>Airfoil chord</td>
<td>[m]</td>
</tr>
<tr>
<td>$U_\infty$</td>
<td>Free stream velocity</td>
<td>[m/s]</td>
</tr>
<tr>
<td>T</td>
<td>Temperature</td>
<td>[K]</td>
</tr>
<tr>
<td>R, r</td>
<td>Radius</td>
<td>[m]</td>
</tr>
<tr>
<td>d</td>
<td>Diameter</td>
<td>[m]</td>
</tr>
<tr>
<td>p</td>
<td>Pressure</td>
<td>[Pa]</td>
</tr>
<tr>
<td>q</td>
<td>Dynamic pressure</td>
<td>[Pa]</td>
</tr>
<tr>
<td>$p_\infty$</td>
<td>Free stream pressure</td>
<td>[Pa]</td>
</tr>
<tr>
<td>$q_\infty$</td>
<td>Free stream dynamic pressure</td>
<td>[Pa]</td>
</tr>
<tr>
<td>M</td>
<td>Mach number or moment</td>
<td>[-] [Nm]</td>
</tr>
<tr>
<td>E</td>
<td>Total energy</td>
<td>[J]</td>
</tr>
<tr>
<td>a</td>
<td>Speed of sound</td>
<td>[m/s]</td>
</tr>
<tr>
<td>$S_{ij}$</td>
<td>Stress tensor</td>
<td>[s$^{-1}$]</td>
</tr>
<tr>
<td>R</td>
<td>Gas constant</td>
<td>[J/(kg K)]</td>
</tr>
<tr>
<td>D</td>
<td>Drag</td>
<td>[N]</td>
</tr>
<tr>
<td>$c_v$, $c_p$</td>
<td>Specific heat at constant volume</td>
<td>[J/kg K]</td>
</tr>
<tr>
<td>M</td>
<td>Mass</td>
<td>[kg]</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
<td>[-]</td>
</tr>
<tr>
<td>H</td>
<td>Boundary layer shape factor</td>
<td>[-]</td>
</tr>
<tr>
<td>$C_f$</td>
<td>Friction coefficient</td>
<td>[-]</td>
</tr>
<tr>
<td>i</td>
<td>Base vector in x-direction</td>
<td></td>
</tr>
<tr>
<td>j</td>
<td>Base vector in y-direction</td>
<td></td>
</tr>
<tr>
<td>k</td>
<td>Base vector in z-direction</td>
<td></td>
</tr>
</tbody>
</table>

Greek letters

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$</td>
<td>Angle of attack</td>
<td>[$^\circ$]</td>
</tr>
<tr>
<td>$\beta$</td>
<td>Sideslip angel</td>
<td>[$^\circ$]</td>
</tr>
<tr>
<td>$\delta$</td>
<td>Boundary layer thickness</td>
<td>[m]</td>
</tr>
<tr>
<td>$\delta^*$</td>
<td>Displacement thickness</td>
<td>[m]</td>
</tr>
<tr>
<td>$\delta_R$</td>
<td>Maximum body diameter</td>
<td>[m]</td>
</tr>
<tr>
<td>$\delta_{ij}$</td>
<td>Kronecker delta</td>
<td>[-]</td>
</tr>
<tr>
<td>$\theta$</td>
<td>Boundary layer momentum thickness</td>
<td>[-]</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>Pressure gradient parameter</td>
<td>[-]</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
<td>[kg/m$^3$]</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Dynamic viscosity</td>
<td>[kg/(ms)]</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity</td>
<td>[m$^2$/s]</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Shear stress</td>
<td>[N/m$^2$]</td>
</tr>
<tr>
<td>$\Phi$, $\varphi$, $\phi$</td>
<td>Velocity potential function (General, outer region, inner region)</td>
<td>[-]</td>
</tr>
</tbody>
</table>

Other notations used are described in the text.
1 INTRODUCTION

1.1 Background

When designing an aircraft it is important in the early design phase to be able to make predictions of the forces and moments acting on the vehicle due to the surrounding airflow. In the beginning of a project a very simplified geometry is often used. Wings, body etc. are then given by flat surfaces which describe the plan form. The aerodynamic analyse tools therefore have to be adjusted to handle these simple geometries and deliver results for a large number of configurations in a short time i.e. seconds/minutes. To meet these requirements the modelling is simplified by assuming that the flow field velocity can be described by the gradient of a potential. To get a unique flow field a wake has to be introduced behind the aircraft. This results in a linear partial differential equation that can be solved by a so called panel method, today there exists a lot of different computation programs using this technique, each with its own disadvantages and advantages.

Panel programs can be subdivided into two groups: low order and high order. In a low order panel method, singularities are distributed with constant strength over each panel. In a higher order method, singularity strengths are allowed to vary linearly or quadratic over each panel. Hence the higher order panel method gives better accuracy in the modelling of the flow field, but this is at the expense of increased code complexity and computation time. Today Computer resources are increasing for every day and therefore it is now possible to use higher order panel methods and still obtain results fast.

A potential flow panel program is often divided into one main part and several sub models. The main program computes numerically the flow field i.e. the velocity potential around a three dimensional body. In this part the flow is treated as inviscid, incompressible and irrotational. When the main program has delivered a solution of the flow field the sub models are used to compute flow field properties such as velocity. From the velocity it is then possible to deduce the pressure distribution which later is used to compute other properties as for example forces and moments acting on the aircraft body.

Most of the panel programs today are coupled to a boundary layer model which adds a viscous contribution. The boundary layer model itself is divided into two parts, one for laminar and one for turbulent boundary layer analysis. Some programs also have a model treating transition and flow separation.

1.2 Objectives

The objectives of this work are further development of an already existing panel program called ADAPDT (Aero Dynamic Analysis and Preliminary Design Tool) developed at Saab Aerosystems for potential flow calculations around aircraft. For more details about this program and the methods used see (Reference 1-2). In this thesis two models are going to be developed, one body model that treats flow around three dimensional slender bodies by using a panel method and a boundary layer model which can give a preliminary estimate of boundary layer properties. Both these models should not increase the calculation time significantly.
1.3 Limitations

The development of these models is limited to the use of Matlab as programming language, since the original program uses it and that Matlab has some advantages compared to other language such as C or FORTRAN.

1.4 Reading Instructions

The report is presented in a way that a person with a Master of Science should be able to comprehend the information. Sections 2.2 and 3.1 give an introduction to the different methods studied. A person with knowledge in aerodynamics can neglect these and continue to parts 2.3 and 3.2 where the theory used in the different models is outlined. The results from the two contributions are given in sections 2.5 (Body model) and 3.6 (Boundary layer model). In the appendices there are more details about the different test cases that have been studied and in the very end there are some figures illustrating the resulting models.

References in the text are given as (Reference 1-15).

References to appendices are denoted as [Appendix A-E].


\section*{Body Model}

\subsection*{Problem Description and Requirements of Body Contribution}

The body flow model developed in this thesis is going to be part of a fully capable potential flow computation program developed at Saab in Linköping described in (Reference 1-2). The program as it is today consists of a pre-processing part, a solver and a post-processing part. The pre-processing part handles the geometry using flat plates defined by four coordinates, one for each corner. These plan forms defines wings, flaps and bodies, the surfaces will be divided into panels by use of a mesh generating program. Every panel will be associated with data such as its position, control point (surface midpoint) coordinates in which the flow velocity are going to be calculated. This way of modelling the aircraft body works quite well as long as the bodies are flat and are exposed to flow in the x-z plane (no sideslip). But most of the world’s civil aircraft have bodies that are slender and circular. When making predictions of the aircraft dynamic stability results from panel programs with enough accuracy even with side slip are needed to be able to calculate the stability derivatives for the aircraft. Another problem with many of the existing panel programs are that they often are written several years ago without any description of the calculation steps used and the implemented code.

The aim of this body model is thus to develop a program which can treat any three dimensional slender body with a panel method that can be used in early design phases with the same short calculation time as the panel methods already existing, for example the NASA Wing-Body program. But also create a program with good documentation so that further development and maintenance is possible.

The requirements of this body contribution are summarized below:

\begin{itemize}
  \item Make calculations of the flow field around a three dimensional body by use of a panel method. This includes calculation of the velocity potential and the flow velocity at the body surface.
  \item Use of a method that can be connected to the already existing program.
  \item Obtain results with enough accuracy to be able to determine stability derivatives when the body is exposed to both sideslip and angle of attack.
  \item Use of the same simple geometry as the present program do.
  \item The body contribution shall be computed in such a way that calculation time not increases.
  \item The program shall be written in Matlab to ensure both easy maintenance and further development of the program in the future.
\end{itemize}
2.2 Theoretical background

2.2.1 General Flow Theory

The flow around a general shaped body moving in a fluid can be described by Navier-Stokes equations. These equations treat all fluid flows and can take care of both viscous phenomena and turbulence. Navier-Stokes equations consist of four equations and can be found in (Reference 3-4).

\[ \frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} (\mu S_{ij}) \quad \text{Eq. 2.1} \]

Continuity

\[ \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0 \quad \text{Eq. 2.2} \]

Energy

\[ \frac{\partial}{\partial t} (\rho E) + \frac{\partial}{\partial x_j} ((\rho E + p) u_j) = - \frac{\partial}{\partial x_j} \left( -k \frac{\partial T}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left( u_i \mu S_{ij} \right) \quad \text{Eq. 2.3} \]

with \( E = \frac{1}{2} u_i^2 + \frac{p}{(\gamma - 1) \rho} \)

Gas law

\[ p = \rho RT \quad \text{Eq. 2.4} \]

The stress tensor \( S_{ij} \) is defined as:

\[ S_{ij} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \quad \text{Eq. 2.5} \]

The first three equations describe the conservation of mass and energy. In the tensor formulation the Einstein tensor summation and derivation is used. Since the Navier-Stokes equations are non-linear partial differential equations they are difficult to solve, especially for high Reynolds numbers. To be able to solve the equations they are simplified, this is done by restricting the calculations to an inviscid fluid. In reality there are of course no inviscid fluid, but for flows with high Reynolds number it can be a reasonable approximation to treat the fluid as inviscid and that the viscous forces are restricted to a small layer close to the body surface. With the approximation above the Navier-Stokes equations are simplified to the Euler equations.

\[ \frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = - \frac{\partial p}{\partial x_i} \quad \text{Eq. 2.6} \]
\[
\begin{align*}
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) &= 0 \\
\frac{D}{Dt} (\rho E) + \frac{\partial}{\partial x_j} ((\rho E + p) u_j) &= 0
\end{align*}
\]

Eq. 2.7
Eq. 2.8

With the gas law repeated

\[p = \rho RT\]

These equations are solved together with some appropriate boundary conditions. The physical boundary condition for Navier-Stokes equations would be a no-slip condition on a solid wall, caused by viscous effects. But since the Euler equations do not consider viscous effects there will be a boundary condition, which allows tangential velocity at the body surface.

### 2.2.2 Potential Theory

In potential theory the flow representations are simplified even more. In addition to the inviscid simplification the models used in this thesis are all based on the assumption that the flow is irrotational. This implies that the fluid element have no angular velocity; rather, their motion through space is a pure translation. Irrotational flow is defined as a flow where the vorticity is zero at every point, i.e.

\[\nabla \times V = 0\]

where \(\nabla\) is the nabla operator. The velocity can then locally be written

\[V = \nabla \phi\]

Eq. 2.9
Eq. 2.10

It is further assumed that the flow is isentropic and steady Eq. 2.10 inserted in Eq. 2.7 yields

\[
\left(1 - \frac{\Phi_x}{a^2}\right) \Phi_{xx} + \left(1 - \frac{\Phi_y}{a^2}\right) \Phi_{yy} + \left(1 - \frac{\Phi_z}{a^2}\right) \Phi_{zz} - 2 \frac{\Phi_x}{a^2} \Phi_x - 2 \frac{\Phi_y}{a^2} \Phi_y - 2 \frac{\Phi_z}{a^2} \Phi_z = 0
\]

Eq. 2.11

with:

\[a^2 = a_0^2 - \frac{\gamma - 1}{2} (\Phi_x^2 + \Phi_y^2 + \Phi_z^2)\]

Where \(a_0\) is the stagnation (or total) speed of sound.

This equation is a nonlinear exact partial differential equation. To be able to make easy calculations the assumption of small perturbations are made, and the equation is linearized. It is also necessary that the body is considered thin. Due to small density variation the flow can be treated as incompressible. This gives the linearized perturbation velocity potential equation according to:

\[
(1 - M^2) \Phi_{xx} + \Phi_{yy} + \Phi_{zz} = 0
\]

Eq. 2.12
The boundary condition at a solid wall and at infinite distance away from the body used to solve Eq. 2.12 can be written

\[ \frac{\partial \phi}{\partial n} = 0 \quad \text{i.e. no fluid through the wall} \]
\[ |\nabla \phi| \to 0, |x| \to \infty \quad \text{i.e. free stream properties} \]

In order to get correct lift around the body a Kutta condition need to be imposed at the trailing edge. This can be done using a wake behind the body; the condition on the wake boundary is that the pressure should be continuous over the wake.

### 2.2.3 Elementary Flow Cases

If the potential can be determined by use of Eq. 2.12 the velocity field can easily be calculated. For a general shaped body it is difficult to calculate the potential therefore a finite number of elementary flow cases are used to approximate the flow field. There are four different elementary flows: uniform, source-sink, doublet and vortex flow. Source-sink flow is a singularity in the flow where the fluid flow moves out from or into the origin. A doublet occurs when a source and a sink with equal strength are moved infinitely close to each other so that the streamlines consists of circles whose centre is on the y-axis. Vortex flow has an infinite velocity at the centre and the fluid particles rotate around the centre.

![Elementary flows](image)

**Figure 1: Elementary flows**
The potential expressions for the different flows in two dimensions are given by Eq. 2.13-Eq. 2.16.

**Uniform flow:** \[ \phi = V_\infty \cdot x \] 
Eq. 2.13

**Source and sink:** \[ \phi = \frac{\Lambda}{2\pi} \ln(r) \] 
Eq. 2.14

**Irrotational vortex:** \[ \phi = \frac{-\Gamma}{2\pi} \theta \] 
Eq. 2.15

**Doublet:** \[ \phi = \frac{k \cdot \cos \theta}{2\pi \cdot r} \] 
Eq. 2.16

Here \( \Lambda \) is the strength of the source, \( k \) and \( \Gamma \) is the strength of the vortex respectively the doublet. These elementary solutions can now be superimposed to find solutions to the linearized potential equation. In three dimensions the source becomes a line source i.e. infinitely number of sources located on a line. In the same manner the vortex becomes a vortex sheet.

### 2.2.4 Panel Methods

Panel methods are approximated solutions of potential theory which with advantage can be solved on a computer. The body is build up by panels, each with is own elementary solution connected to itself. The elementary solutions are superimposed with different strength of each elementary solution so that they together fulfil the boundary conditions. The most common way to create a panel program is by using vortex panels for the lifting surfaces such as the wings because with help of a vortex sheet it is possible to calculate the lift which the wing generates. Around non lifting bodies it is common to use a source panel method. Today this technique has become a standard aerodynamic tool in the industry. In fact, numerical solutions of potential flow by both source and vortex panel techniques have revolutionized the analysis of low-speed aerodynamics.

![Figure 2: The panel method applied on an entire aircraft](image-url)
2.3 Method

In this case, the flow field around a three dimensional body is assumed to be inviscid, irrotational, and incompressible and slender. Instead of using one flow field around the body and use of three dimensional panels along the body surface, which is most common among panel programs, the flow field around the aircraft body is divided into two regions as shown in Figure 3, one inner region closest to the body and one outer region which reaches from the inner regions outer boundary to infinity. The problem is solved by dividing the body into several cross sections lengthwise i.e. from \( x = 0 \) to \( x = L \) where \( L \) is the body’s length. In each of these cross sections the flow will be treated as a two dimensional problem where the inner and outer boundary are split up in panels. Treating the inner flow field as two dimensional is of course an approximation to achieve a reduced computation time. The velocity potential \( \phi \) in the inner region is calculated by use of boundary conditions along the body surface. The outer problem is where the three dimensional connection will be obtained. The result from the inner problem is connected to the three dimensional panels of which the outer problem consists. When the outer problem is solved the solution will be transferred to the inner problem through boundary conditions and the iteration process starts over and continues until convergence has occurred. Figure 3 below shows one example of a cross section with inner and outer boundary. Observe that in this case the outer boundary has the shape of a cylinder but this is of course not necessary, a square or rectangular box will do as well.

2.3.1 The Inner Problem

Figure 3 gives an example of an inner problem with circular cylinder as outer boundary.

\[
\phi_{yy} + \phi_{zz} = 0 \quad \text{Eq. 2.17}
\]

![Figure 3: Example of an inner problem.](image-url)
For a more detailed description of the equation in two dimensions see Ashley & Landahl (Reference 3). Outside of \( \Omega \) it is instead Laplace equation in three dimensions that will be used to describe the flow but this will be explained later in this chapter. The inner problem is connected to the outer problem along the boundary \( \Gamma_2 \) according to

\[
\phi_{\Gamma_2} = \phi_{\Gamma_2}
\]

Eq. 2.18

Where \( \phi \) is the potential in the inner area and \( \varphi \) in the outer.

The solution in the inner region is written:

\[
\phi^i = \frac{\int_{\Gamma_1}}{r_1} + \frac{\int_{\Gamma_2}}{r_2}
\]

Eq. 2.19

Where the integrals are written as finite sum of integrals over the panels \( Li \)

\[
\int_{\Gamma_1} = \sum_{i=1}^{N} \frac{1}{2\pi} \int_{L_i} \left( -\frac{\partial \phi}{\partial n} \ln|\rho| + \phi \frac{\partial}{\partial n} \ln|\rho| \right) ds
\]

Eq. 2.20

\[
\int_{\Gamma_2} = \sum_{i=1}^{M} \frac{1}{2\pi} \int_{L_i} \left( -\frac{\partial \phi}{\partial n} \ln|\rho| + \phi \frac{\partial}{\partial n} \ln|\rho| \right) ds
\]

Eq. 2.21

\( n \) is a normal vector which will be defined later.

Eq. 2.20 and Eq. 2.21 expresses the velocity potential in an arbitrary point in \( \Omega \). The equation can be derived from Green’s formula applied in two dimensions.

Where \( \rho = \sqrt{(y - y_1)^2 + (z - z_1)^2} \)

Eq. 2.22

The coordinate \( (y_1, z_1) \) corresponds to the control point in which the potential \( \phi \) is evaluated.

Addition of equations Eq. 2.20 and Eq. 2.21 gives:

\[
\phi_j = \sum_{i=1}^{N} \frac{1}{2\pi} \int_{L_i} \left( -\frac{\partial \phi}{\partial n_i} \ln|\rho| + \phi \frac{\partial}{\partial n_i} \ln|\rho| \right) ds + \sum_{i=1}^{M} \frac{1}{2\pi} \int_{L_i} \left( -\frac{\partial \phi}{\partial n_i} \ln|\rho| + \phi \frac{\partial}{\partial n_i} \ln|\rho| \right) ds
\]

Eq. 2.23

Where

\[
\phi_j \approx \phi(y_j, z_j)
\]
performed so that the normal vector to points out from $\Omega$ i.e. that the normal vector is directed against the body centre along the inner boundary and away from the body centre on the outer boundary.

Now the equation for each panel $\Gamma_i$ can be expressed as:

$$\Gamma_i: \{(y, z) = (y_i, z_i) + p \cdot \bar{r}\}$$  \hspace{1cm} \text{Eq. 2.24}

$$\bar{r} = (y_i, z_i) - (y_{i+1}, z_{i+1})$$  \hspace{1cm} \text{Eq. 2.25}

$p$ gives relative position on the current panel.

To be able to solve the integrals in equation Eq. 2.23 the following rewriting has to be done:

$$\frac{\partial}{\partial n_i} \ln \rho = \bar{n}_i \left( \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) \ln \rho$$  \hspace{1cm} \text{Eq. 2.26}

The normal vector $\bar{n}$ can be written:

$$\bar{n} = \frac{(\Delta z, -\Delta y)}{\Delta s} = \frac{(\Delta z, -\Delta y)}{\sqrt{(\Delta y)^2 + (\Delta z)^2}}$$  \hspace{1cm} \text{Eq. 2.27}

$$ds = |\bar{r}| dp$$  \hspace{1cm} \text{Eq. 2.28}

where

$$|\bar{r}| = \sqrt{\Delta y^2 + \Delta z^2}$$  \hspace{1cm} \text{Eq. 2.29}

Substitution of Eq. 2.26-2.29 into Eq. 2.23 gives two equations, one for each boundary.
- Body model -

Outer boundary:

\[
\phi_{o,j} = \sum_{i=1}^{N} \frac{\phi_{i,i}}{2\pi} \int_{0}^{1} \bar{n}_{i} \left( \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) \ln|\rho| \, dp - \sum_{i=1}^{N} \frac{1}{2\pi} \left( \frac{\partial \phi_{i}}{\partial n} \right) \int_{0}^{1} \left[ \ln|\rho| \right] \, dp
\]

\[
\sum_{i=1}^{N} \frac{\phi_{o,i}}{2\pi} \int_{0}^{1} \bar{n}_{i} \left( \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) \ln|\rho| \, dp - \sum_{i=1}^{N} \frac{1}{2\pi} \left( \frac{\partial \phi_{i}}{\partial n} \right) \int_{0}^{1} \left[ \ln|\rho| \right] \, dp
\]

Eq. 2.30

Inner boundary:

\[
\phi_{i,j} = \sum_{i=1}^{N} \frac{\phi_{i,i}}{2\pi} \int_{0}^{1} \bar{n}_{i} \left( \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) \ln|\rho| \, dp - \sum_{i=1}^{N} \frac{1}{2\pi} \left( \frac{\partial \phi_{i}}{\partial n} \right) \int_{0}^{1} \left[ \ln|\rho| \right] \, dp
\]

\[
\sum_{i=1}^{N} \frac{\phi_{o,i}}{2\pi} \int_{0}^{1} \bar{n}_{i} \left( \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) \ln|\rho| \, dp - \sum_{i=1}^{N} \frac{1}{2\pi} \left( \frac{\partial \phi_{i}}{\partial n} \right) \int_{0}^{1} \left[ \ln|\rho| \right] \, dp
\]

Eq. 2.31

Index o corresponds to the outer boundary and i the inner boundary. These two equations form a linear equation system

\[
\begin{pmatrix}
A_{11} & A_{12} \\
A_{21} & A_{22}
\end{pmatrix}
\begin{pmatrix}
\phi_{i} \\
\frac{\partial \phi}{\partial n}
\end{pmatrix}
= \begin{pmatrix}
B_{1} \\
B_{2}
\end{pmatrix}
\]

\[
\iff A \cdot \Phi = B
\]

Eq. 2.32

Where the boundary conditions on the inner and outer boundary inserted in Eq. 2.30 and Eq. 2.31 resulting in the right hand side of Eq. 2.32.

Further derivation of the coefficients in matrix A and vector B can be seen in [Appendix D]. A is a quadratic matrix with N + M rows and columns, B and \( \Phi \) are two column vectors with N + M elements. The equation system above can easily be solved with Matlab or any other mathematical computer program and the matrix \( \Phi \) can be obtained.

### 2.3.2 Calculation of the Velocity at the Body Surface

When the potential \( \phi_{i} \) is known at the inner boundary the flow speed can be computed in the control points by use of equation Eq. 2.32. The velocity on the inner boundary can be obtained by differentiating Eq. 2.31 with the solution of \( \phi_{i} \) and \( \left( \frac{\partial \phi}{\partial n} \right) \) from Eq. 2.32 inserted.

This gives

\[
V_{i} = C_{11} \cdot \phi_{i} + C_{12} \cdot \left( \frac{\partial \phi}{\partial n} \right)_{i} + C_{21} \phi_{o} + C_{22} \left( \frac{\partial \phi}{\partial n} \right)_{o}
\]

Eq. 2.33

Derivation of the coefficients can be seen in [Appendix D].
The equation above expresses the perturbation velocity at the body surface in the cross section components y and z. This velocity is now going to be added to the free stream velocity which implies that the x-component will be the dominating part.

\[ U = U_\infty (x, y, z) + V_i \]  
\[ \text{Eq. 2.34} \]

2.3.3 Velocity Contribution due to Vortices

When a wing generates lift there are cortices formed by the wing. Vortices are developed because the pressure is higher below the wing than above which makes the air moving from the lower side to the top side of the wing. This gives a rolling motion into the air which then forms vortices. These vortices have to be taken into account when calculating the velocity at all points situated behind the lifting surface which created the vortex. This vortex contribution will be added to the free stream velocity when the boundary condition at the body surface is evaluated. The velocities induced by the vortex are denoted \( \vec{u}_v \) and are included in the following boundary condition calculations. Which vortex that affects a certain node point is kept together by a small program already implemented in the outer flow calculation and will not be described further.

2.3.4 Calculation of Boundary Condition at the Body Surface

At the surface of the body the summation of the free stream and the perturbation velocity has to be zero. This can be expressed as follows

\[ \vec{u} \cdot \vec{n} = 0 \iff n_y \frac{\partial \phi}{\partial y} + n_z \frac{\partial \phi}{\partial z} = -U_\infty \cdot \vec{n} - \vec{u}_st \cdot \vec{n} \]  
\[ \text{Eq. 2.35} \]

If Eq. 2.35 is going to be satisfied then the boundary condition at the inner boundary has to be (with the left hand side rewritten)

\[ \frac{\partial \phi}{\partial n} = -U_\infty \cdot \vec{n} - \vec{u}_st \cdot \vec{n} \]  
\[ \text{Eq. 2.36} \]

Here the normal vector is directed outwards from the body surface.

2.3.5 The Outer Problem

To this part of the problem there already exist an implemented computation model but the major parts are as follows.
In the outer flow region the flow is given by the equation:

\[ (1 - M^2) \rho_{xx} + \rho_{yy} + \rho_{zz} = 0 \]  \hspace{1cm} \text{Eq. 2.37}

On the boundary between the inner and outer flow area is the following condition valid

\[ \frac{\partial \phi}{\partial n} = \frac{\partial \phi}{\partial n} \]  \hspace{1cm} \text{Eq. 2.38}

This condition connects the outer with the inner problem. If for example a wing is connected to the aircraft which also affects the outer problem there is a boundary condition according to

\[ \frac{\partial \phi}{\partial n} = 0 \]  \hspace{1cm} \text{Eq. 2.39}

along the boundary of the wing.

With help from the boundary conditions Eq. 2.38 can be solved and the solution is sent back to the inner problem and a new iteration takes place. This continues until the iteration process has converged.

### 2.3.6 Calculation of the Pressure Distribution \( C_p \)

The pressure around the body can be derived easily by use of the earlier obtained flow velocity around the body.

From Anderson (Reference 4)

\[ C_p = \frac{p - p_{\infty}}{q_{\infty}} = 1 - \left( \frac{U}{U_{\infty}} \right)^2 \]  \hspace{1cm} \text{Eq. 2.40}

Here \( U \) is the total velocity (disturbance + free stream velocity).

Schlichting (Reference 5) presents an alternative equation which can be derived from Eq. 2.40.

\[ C_p = \left[ 2 \cdot \frac{u}{U_{\infty}} \left( \frac{u}{U_{\infty}} \right)^2 + \left( \frac{w}{U_{\infty}} \right)^2 \right] \]  \hspace{1cm} \text{Eq. 2.41}

Here \( u \) is the disturbance velocity in the x direction and \( w \) is velocity in the radial direction. Note that in Eq. 2.41 the two last terms can be neglected because they are small compared to the first one especially if the body is exposed to axial flow.
$w_r$ is computed directly from the two dimensional problem whereas the x disturbance velocity is obtained by differentiating the potential $\phi$ on the body surface. The disturbance velocity can be written:

$$u_x = \frac{\partial \phi}{\partial x} \quad \text{Eq. 2.42}$$

This differential can now be solved by using a numeric derivation (central difference) along the body surface.

Let $g(x) = \phi(x, y(x), z(x))$, this gives

$$u_x = \frac{g(x + h) - g(x - h)}{2h} - \frac{dy}{dx} \cdot u_y - \frac{dz}{dx} \cdot u_z \quad \text{Eq. 2.43}$$

Here $h$ is the distance between the grid points in the x-direction. Since $\phi$ is depending on $y$ and $z$ as well this equation gives two unwanted contributions from the derivatives with respect to the $y$ and $z$ directions. Therefore these terms have to be subtracted according to Eq. 2.43. These two derivatives are also derived by use of numeric differentiation.

### 2.4 Data Management

In this thesis Matlab is used as programming language mostly since the present program used it. But that is not the only reason, Matlab have several advantages compared other programming languages. Since the equations and the numerical methods described in the theory section mostly deals with matrix assembly and solving linear and nonlinear equation systems Matlab was superior. The purpose when developing Matlab was to create a program that handles matrix and vector calculations in a fast and effective manner. Matlab also have advantages when it comes to maintenance compared to other programming languages as FORTRAN.

The programming can be divided into three different parts. These parts are the geometry input, calculation of the flow in the inner region and the connection between the inner- and the outer solution and here follows a short description of the program.

#### 2.4.1 Program Structure

The program is based on seven different Matlab m-files with the following names:

- body_start.m
- body_solve.m
- eqsolve.m
- twod.m
cocalc.m
velocity_calc.m
secdev.m

The first file (body_start) is used to prepare all input variables to be used in the panel programs solver. This is due to that the outer program makes all calculations of the velocity potential $\phi$ by use of three dimensional panels while the inner problem makes calculations in two dimensions at every x-station. This implies that all two dimensional variables has to be converted into three dimensions when going from the inner problem to the outer problem or the other way around. These conversions are made by interpolation in the x- direction of the two dimensional panel contributions to the control points of the three dimensional panels and the reversed process when going from 3D to 2D. In the latter case it is the known potential $\phi_o$ at the outer boundary that is converted whereas in the other case around it is $\frac{\partial \phi}{\partial n}$ on the outer boundary.

The next file (body_solve.m) loops the solver of the two dimensional problem for every cross section. The rest of the files are part of the solver that makes calculations for each cross section. Function twod.m performs the different summations over the outer respectively the inner boundary when computing the potential using cocalc.m which contains the derivatives of first degree. The function velocity_calc.m calculates the velocity at the body surface trough use of the derivatives of second degree which are stored in file secdev.m.

2.4.2 Geometry Treatment

There are some different cases of geometries which also have to be accounted for in the main program. For example what happens if there exists a T-tail or a canard configuration where most of these surfaces are going to be included in the inner problem?, or why not an aircraft with two-four engines external which almost every airliner has to day. These engines will also be treated as bodies and the two dimensional theory need to be applied around those as well. From this point of view the aircraft can consist of an infinite number of bodies which also the panel program has to take care of.

In order to make the program handling an aircraft structure consisting of more than one body cell arrays instead of ordinary matrix variables have been used. The output variables are instead made consisting of one cell corresponding to each body.

The next case is the sections where the body has for example wings, tail surfaces and so on. These are easily taken care of as long they can be treated as one object which means that the inner boundary can be connected all the way around by only one curve that is simply connected. When this is true the calculation are solved as usual with the only difference that there probably exists some more panels along the inner boundary to achieve better accuracy. In Figure 5 below the flow calculation for a cross section containing a T-tail can be seen, note that the different parts of the geometry are simply connected.
Figure 5: Body section with T-tail configuration.

It is when the geometry can not be connected to each other without disturbing the flow there is a small difference in the solution method. This occurs for example for an aircraft with swept wings where the wingtips are further to the rear than the inner trailing edge of the wing. An example of this kind of configuration can be seen in Figure 6.

Figure 6: Section containing more than one element

To solve this, a program that loops over more than one element when the summation over the inner boundary was developed. But there will still be one system of equations to solve because
when the program performs the summation it starts with element one which for example might use position 1-10 in the coefficient matrix and then it continues to the next element and uses position 11 and forward until the contribution from all elements have been accounted for. Observe that the boundary condition according Eq. 2.36 is valid along all elements in the current section.

2.5 Summary of Computation Results Using the Body Model

The resulting body model/contribution developed in this thesis have been validated by performing calculations around some bodies to which it was possible to find analytical solutions, resulting data from numerical techniques such as Euler or Navier-Stokes calculations or experimental results from for example wind tunnel tests. The validations performed with the body model can be found in [Appendix A].

The purpose with the validation was to find the solution accuracy and method limitation. In the first validation example the accuracy of the body model was checked by comparing the two dimensional potential \( \phi \) in the inner region using contour plots. The resulting contour plots can be seen in Figure 13-16. The calculations were performed on two different cross sections, one with wings inside the reference box and one without. The results of these calculations shows excellent agreement between the analytical solution and the numerical panel method.

In the second validation calculations on an entire three dimensional body is performed. A parabolic body and a elliptic body of revolution are used. The reason for this is that there exists analytical solutions for them. The pressure distribution along the bodies have been compared to analytical solutions. To find some of the body models limitations there are several calculations made with different number of panels, body dimensions and size of the reference box. The computed pressure distributions over the two bodies can be seen in Figure 17-18.

The results from these validations are:

- The number of panels in each cross section needed when calculations are performed on a circular geometry are, 10 around the body cross section and 20 on the outer reference box. This configuration gives results that deviates less than 2 % from the analytical solution. From the validations the recommended numbers of panels for most types of sections are 20 along the inner and 40 along the outer boundary. If there exists wings or tail surfaces it is necessary to use at least 50 panels around the inner boundary.

- The number of cross sections needed for the program to provide an error less than one percent is 100 for the geometries in the second validation [Appendix A].

- The results are not sensitive to the outer box size but the preferable size is \( 1.4 \cdot r_{\text{body, max}} \).

- The maximum slenderness ratio of the body could not be greater than 1/5.

- The difference in the results between a square box as outer reference and a circular cylinder was negligible, because of simplicty a rectangular box have been used.
- Body model -

- The calculation time for the inner solution area with the above configuration is less than 15 seconds for each body.

From the second validation it is also noticeable that the panel method does not work well at the ends of elliptic shaped bodies. This is due to the derivates of the normal vector which approaches infinity at the ends.

Some resulting calculation examples/aircraft geometries that can be treated in the final panel program can be seen in [Appendix D]. Observe that all of these aircrafts are fakes and do not exist in reality. Figure 29 gives an example of a two engine aircraft with a swept wing and a V-tail. In this case the engines are also treated as bodies. Three reference boxes are then used, one for each body. Note also the simple geometry used to create the different aircrafts, all points used can with ease be taken from for example a CAD program.

2.5.1 Discussion of Body Model

The resulting body model developed in this thesis gives excellent results in the validations performed but it is important to remember that a panel program for potential flow is always an approximation and that, in this case, it is depending on the boundary conditions at the outer reference box. The results are not reliable at high angle of attacks, say about 15-20° since the model do not treat separated flow. As can be seen in the calculations around the three dimensional bodies the model gives inaccurate results at the ends of a non sharp body. This is a well known problem with this type of panel method and one way to solve it might be some kind of analytical solution that are used in the cross sections closest to the nose and the rear point.

The number of panels needed to give accurate results can seem quite low but it is worth mentioning that if the number of panels is doubled the calculation time will be significantly higher, and time is very valuable especially in the early design phases where this program is intended to be used.

The panel formulation used here is simple and seems to give good results but the accuracy of the model can be improved if the potential is allowed to vary linearly or quadratic over each panel. Disadvantages with this are that the problem formulation and the code will be more complicated which implies longer computation time. Therefore it is very important to decide the purpose of the panel program before choosing what method to use.
3 BOUNDARY LAYER MODEL

3.1 Theoretical Background

The boundary layer method developed here consists of a two-dimensional integral method where the contribution from the boundary layer is computed at a number of stations along the wing. In this section some different ways to calculate the contribution which the boundary layer gives to the flow over a two-dimensional wing profile are derived. First the boundary equations are derived and some different approximate solutions are discussed. Two methods treating laminar boundary layer and one treating turbulent boundary layer have been studied. Because of the complexity of a turbulent boundary layer most of the existing methods are quite complicated to apply in reality.

3.1.1 Boundary Layer Theory

The concept of boundary layer theory comes from Prandtl’s hypothesis that

- Adjacent to the solid body in a flow there is a thin layer of fluid within which viscous effect is dominant.
- Outside this layer the viscous effects may be ignored and the flow considered inviscid.

A boundary layer can be laminar or turbulent. The type of this layer depends on the Reynolds number. At low Reynolds number the boundary layer is laminar and when the Reynolds number increases the flow will undergo transition and become turbulent. Even if the layer may be thin its effect is by no means negligible. It is the boundary layer that generates skin friction and the consequent drag. The boundary layer is a dynamic quantity; it grows with the length, leading to an increasing boundary layer thickness \( \delta \). A turbulent boundary layer is different compared to a laminar one, in the sense that an intense mixing in the flow exists. Consequently the skin friction increases in a turbulent boundary layer. Also the boundary layer thickness grows more rapidly in the turbulent case. For a flat plate at zero incidence, the laminar thickness grows as \( \delta \sim 5/\sqrt{\text{Re}} \) and for the turbulent case \( \delta \sim 0.37/\text{Re}^{0.2} \).

The most common definition of the boundary layer thickness is the distance normal to the surface where the velocity reaches 99\% of the free stream speed.

3.1.2 Boundary Layer Around Airfoils

Flow about an airfoil can be very difficult to predict and the most important effects that contribute to this complexity is transition and separation. Transition is the phenomena of which flow initially laminar becomes turbulent. When this happens depends on the strength in the pressure gradient, the surface roughness and the free stream turbulence. It is evident from experiments that transition does not take place at one point. There is usually a transition region which size depends on the Reynolds number. Today airfoils are made to delay the transition as
much as possible. It is much better in a drag perspective to have the transition point near the trailing edge than the leading edge.

Figure 7: Transition from laminar to turbulent boundary layer

Separation of flow occurs when there is a negative pressure gradient which decreases the flow velocity close to the wall. This leads to a decreasing velocity gradient at the wall. At a certain point this gradient becomes zero implying a zero shear stress. When following the airfoil further downstream the velocity gradient becomes negative which gives a recirculating flow and a so called flow separation. Sometimes laminar separation results in turbulent flow and most of the times the resulting flow is capable of resisting separation and reattach as consequence. See Figure 8 below for illustration.

Figure 8: Laminar separation

If the flow do not reattach then this results in creation of a wake behind the wing/airfoil. The consequence of this is a drastic decrease in lift and a large increase in drag; under such conditions the airfoil is said to be stalled.

3.1.3 The Boundary Layer Equations

The general equations valid for viscous flow, which can be found in Anderson (Reference 4) and Kuethe & Chow (Reference 6), have here been reduced to the boundary layer equations.

The continuity equation for two dimensional steady flows reads;
- Boundary layer model -

\[
\frac{\partial}{\partial x} (\rho u) + \frac{\partial}{\partial z} (\rho w) = 0 \quad \text{Eq. 3.1}
\]

**Momentum equations;**

x-momentum:

\[
\rho u \left( \frac{\partial u}{\partial x} \right) + \rho w \left( \frac{\partial u}{\partial z} \right) = - \frac{\partial p}{\partial x} + \frac{\partial}{\partial z} \left( \mu \frac{\partial u}{\partial z} \right)
\]

z-momentum:

\[
\frac{\partial p}{\partial z} = 0 \quad \text{Eq. 3.2}
\]

**Energy equation;**

\[
\rho u \frac{\partial h}{\partial x} + \rho w \frac{\partial h}{\partial z} = \frac{\partial}{\partial z} \left( k \frac{\partial T}{\partial z} \right) + u \frac{\partial p}{\partial x} + \mu \left( \frac{\partial u}{\partial z} \right)^2 \quad \text{Eq. 3.3}
\]

In Eq. 3.1-3.3 \( p \) is the pressure, \( k \) is the thermal conductivity of the gas, \( \mu \) is the dynamic viscosity, \( h \) the enthalpy and \( T \) is the temperature. \( u \) and \( w \) are the two Cartesian velocity components.

### 3.1.4 Boundary Conditions, Boundary Layer Thickness

At the surface along a flat plate the “no slip” condition is valid. This implies that the relative motion between the surface and the fluid is zero which gives the following boundary conditions:

\[ u = w = 0 \text{ at } z = 0 \quad \text{Eq. 3.4} \]

From Eq. 3.2 at the wall for steady flow and the following can be obtained:

\[
\left( \frac{dp}{dx} \right)_w = \left( \frac{dp}{dx} \right)_e \quad \text{Eq. 3.5}
\]

Where the suffix \( e \) corresponds to the outer edge of the boundary layer.

Since \( \frac{\partial}{\partial z} \left( \mu \frac{\partial u}{\partial z} \right) \approx 0 \Rightarrow \) at the outer edge we get

\[
\left( \frac{dp}{dx} \right)_e = - \rho u_e \frac{du_e}{dx} \quad \text{Eq. 3.6}
\]

which together with Eq. 3.5 yields
- Boundary layer model -

\[
\frac{dp}{dx} = -\rho u_e \frac{du_e}{dx} \quad \text{Eq. 3.7}
\]

Near the wall follows from Eq. 3.2

\[-\frac{dp}{dx} + \frac{\partial}{\partial z} \left( \mu \frac{\partial u}{\partial z} \right) = 0 \quad \text{Eq. 3.8}
\]

which combined with Eq. 3.7 results in

\[
\left( \frac{\partial^2 u}{\partial z^2} \right)_w = -u_e \frac{du_e}{dx} \quad \text{Eq. 3.9}
\]

This equation will be used in the derivation of the momentum integral equation. At the outer edge of the boundary layer, the following quantities are defined

\[u = u_e, \quad \rho = \rho_e \quad \text{at} \quad z = \delta \quad \text{Eq. 3.10}\]

\(\delta\) is the boundary layer thickness which is a small value at high Reynolds numbers. Two other quantities of physical significance in boundary layer theory are defined as follows.

Displacement thickness

\[\delta^* = \int_0^h \left( 1 - \frac{\rho u}{\rho_e u_e} \right) dz \quad \text{Eq. 3.11}\]

Where h is the distance in Figure 10. The displacement thickness is the distance the fluid is displaced out from the boundary into the external flow. See Figure 9. The effect on the external flow from the boundary layer is the same as if the whole body surface is displaced a distance \(\delta^*\).

![Figure 9: Displacement thickness](image-url)
Momentum thickness

\[ \theta = \int_0^h \frac{\rho u}{\rho_e u_e} \left( 1 - \frac{u}{u_e} \right) dz \tag{Eq. 3.12} \]

\( \theta \) is a parameter proportional to the decrement in momentum flow due the presence of the boundary layer.

Shape factor

The shape factor is defined as the ratio

\[ H = \frac{\delta^*}{\theta} \tag{Eq. 3.13} \]

Momentum integral equation

By integrating Eq. 3.2 with respect of \( z \) from \( z = 0 \) to the outer edge of the boundary layer an equation describing the overall rate of flux of momentum across a cross section of the boundary layer depending on the local pressure gradient and surface frictional stress can be obtained. This so called momentum integral equation is used as the basis of a class of approximate methods of solutions.

Imagine two adjacent sections AE and DF of the boundary layer separated with a distance \( \Delta x \). The sections of the boundary layer are assumed to be of unit depth. See Figure 10.

![Figure 10: Boundary layer section used for deriving the momentum integral in two dimensional flow](image)

The rate of mass flow at BC can be calculated according to

\[(\text{Rate of mass flow across section DC}) - (\text{rate of mass flow across AB}):]\]
- Boundary layer model -

\[
\frac{d}{dx}\left[ \int_0^x \rho u dz \right] \Delta x + O(\Delta x^2)
\]

Eq. 3.14

Alternatively the rate of mass flow across BC can be written:

\[
\rho w_h \Delta x
\]

Eq. 3.15

Hence continuity requires

\[
\rho_e w = -\frac{d}{dx}\left[ \int_0^x \rho u dz \right] + O(\Delta x)
\]

Eq. 3.16

Eq. 3.16 multiplied with the velocity at the outer edge gives:

\[
\rho w_h u_e \Delta x = -u_e \frac{d}{dx}\left[ \int_0^x \rho u dz \right] \Delta x + O(\Delta x)^2
\]

Eq. 3.17

Integrate Eq. 3.2 over ABCD results in

\[
\Delta x \frac{d}{dx}\left( \int_0^x \rho u^2 dz \right) + h\Delta x \frac{dp}{dx} + \rho_h w_h \Delta x + \tau_w \Delta x = 0
\]

Let \( \Delta x \to 0 \) then the momentum integral equation can be obtained using Eq. 3.17

\[
\frac{d}{dx}\left( \int_0^x \rho u^2 dz \right) - u_e \frac{d}{dx}\left( \int_0^x \rho u dz \right) = -h \frac{dp}{dx} - \tau_w
\]

Eq. 3.18

This form of the momentum integral equation is not that useful. However it can be expressed in terms of the displacement thickness and momentum thickness by introducing the form factor \( H \). This gives the following equation for incompressible flow. For more details about the derivation see (Reference 7).

\[
\frac{d\theta}{dx} + \frac{1}{u_e} \frac{du_e}{dx} \theta(H + 2) = \frac{\tau_w}{\rho_e u_e^2}
\]

Eq. 3.19

The above equation is called the momentum integral equation and it is used in a variety of different solution methods of boundary layer flows. It is an ordinary differential equation in \( \theta \) and many different solutions have been derived during the years, both for laminar and for turbulent boundary layers. For a turbulent boundary layer some auxiliary conditions to find feasible solutions are needed.

### 3.2 Laminar Solution Methods

As explained earlier there exist a number of different solutions to the boundary layer equations. Here two methods treating laminar and one for turbulent boundary layer are presented.
3.2.1 Thwaites’ Method

Thwaites method for laminar boundary layers is a method based on the momentum integral equation Eq. 3.19. Its merit lies in that it does not need a velocity profile to start with. Thwaite used the boundary layer equation of motion Eq. 3.2 for incompressible flow and defined two parameters:

\[ l = \frac{\theta \left( \frac{\partial u}{\partial z} \right)_w}{u_e \mu} \quad \text{Eq. 3.20} \]

\[ m = \frac{\theta^2 \left( \frac{\partial^2 u}{\partial z^2} \right)_w}{u_e} \quad \text{Eq. 3.21} \]

Through applying Eq. 3.9

\[ m = -\frac{\theta^2}{\nu} \frac{du_e}{dx} \quad \text{Eq. 3.22} \]

Here \( l \) is directly proportional to the frictional stress while \( m \) is related to the external pressure gradient. If these expressions are introduced into the moment integral Eq. 3.19 the following equation is derived

\[ u_e \frac{d(\theta^2)}{dx} = \nu L(m) \quad \text{Eq. 3.23} \]

where \( L(m) = 2[m(\nu + 2) + l] \quad \text{Eq. 3.24} \)

Based on experiments and by calculating velocity profiles Thwaite replaced this equation by a simple linear approximation according to

\[ L(m) = 0.45 + 6m \quad \text{Eq. 3.25} \]

Combining Eq. 3.23 and Eq. 3.25 gives

\[ u_e \frac{d(\theta^2)}{dx} = 0.45\nu - 6\theta^2 \frac{du_e}{dx} \leftrightarrow \frac{d}{dx} \left( u_e^2 \theta^2 \right) = 0.45u_e \theta^2 \quad \text{Eq. 3.26} \]

Hence integrating this equation from the leading edge, \( x = 0 \) to any point \( x = x_1 \) gives
Boundary layer model

\[ \theta^2 - \frac{C}{u_e^6} = 0.45 \frac{\nu}{u_e^6} \int_0^x u_e^5 \, dx \]  \hspace{1cm} \text{Eq. 3.27} \]

where \( C = u_n^6 \cdot \theta^2(0) \). The initial value \( \theta(0) \) will be determined later.

From the parameter \( \theta \) is \( H \) and \( l \) calculated, then follows from the empirical relations

\[ l(m) = 0.22 + 1.57m - 1.8m^2, \quad \text{for} \ 0 < m < 0.1 \]
\[ = 0.22 + 1.402m + \frac{0.018m}{m + 0.107}, \quad \text{for} \ -0.1 < m < 0 \]  \hspace{1cm} \text{Eq. 3.28} \]
\[ H(m) = 2.61 - 0.375m + 5.24m^2, \quad \text{for} \ 0 \leq m \]
\[ = 2.088 + \frac{0.0731}{m + 0.14}, \quad \text{for} \ -0.1 < m < 0 \]  \hspace{1cm} \text{Eq. 3.29} \]

By comparing several different boundary layer flows Thwaite established that in most boundary layers laminar flow separation happens when \( m \) is -0.09 and \( H \) is 3.55. Transition can be predicted with Michel’s criterion but this will be explained later.

Starting conditions for Thwaites method

To solve Eq. 3.27 starting conditions for \( \theta \) are needed, there exists two possible:

- A boundary layer starts at zero thickness as at the leading for a flat plate.
- The boundary layer starts from a stagnation point of the airfoil.

The second one is used because it gives a more accurate result for different types of wings/airfoils that will be used in the Matlab program.

Consider a point close to the leading edge of the wing/airfoil and expanding the velocity in Taylor series, thus

\[ u_e(x) = u_e(0) + \frac{du_e}{dx}(0)x + \frac{1}{2} \frac{du_e^2}{dx^2}(0)x^2 + \ldots \]  \hspace{1cm} \text{Eq. 3.30} \]

Retaining only the first two terms gives

\[ u_e(x) = \hat{u}(0)x = \hat{u}_0x \quad \text{since} \ (u_e(0) = 0) \]  \hspace{1cm} \text{Eq. 3.31} \]

which can be substituted into Eq. 3.26 and simplifying gives

\[ \frac{1}{\nu} \frac{d}{dx} \left( \hat{u}_0^5 x^6 \theta^2 \right) = 0.45 \hat{u}_0^5 x^5 \]

Integrating the above equation yields
\[
\frac{1}{v} \ddot{u}_0 x^6 \theta^2 = 0.45 \ddot{u}_0 x^6 + C
\]

The limit as \( x \) approaches 0 in the left and right hand side gives, \( C = 0 \) hence

\[
\theta(0) = \sqrt{\frac{0.075\nu}{\ddot{u}_0}}
\]

Eq. 3.32

which is the starting value for \( \theta \).

### 3.2.2 Von Karman and Pohlhausen’s Method

This is an early attempt to calculate a boundary layer in presence of a pressure gradient. This method also starts from the momentum integral equation, repeated below

\[
\frac{d\theta}{dx} + \frac{1}{u_e} \frac{du_e}{dx} \theta(H + 2) = \frac{\tau_x}{\rho u_e^2}
\]

With an assumed finite boundary layer thickness \( \delta \), the boundary conditions for the velocity profile at any station are

\[
z = 0, \quad u = 0 \quad \nu \left( \frac{\partial^2 u}{\partial z^2} \right)_w = -u_e \frac{du_e}{dx}
\]

Eq. 3.33

\[
z = \delta, \quad u = u_e \quad \frac{\partial u}{\partial z} = \frac{\partial^2 u}{\partial z^2} = \frac{\partial^3 u}{\partial z^3} = \ldots = 0
\]

Assume that the velocity profile \( u/u_e \) can be expressed as a quartic profile in \( \eta \), where \( \eta = z/\delta \), i.e.

\[
\frac{u}{u_e} = a\eta + b\eta^2 + c\eta^3 + d\eta^4
\]

Eq. 3.34

Then four of the boundary conditions in Eq. 3.33 can be satisfied in addition \( u = 0 \) when \( z = 0 \), in order to determine the coefficients \( a, b, c, d \). By choosing to satisfy:

\[
\eta = 0, \quad \nu \left( \frac{\partial^2 u}{\partial z^2} \right)_w = -u_e \frac{du_e}{dx}
\]

Eq. 3.35

\[
\eta = 1, \quad u/u_e = 1, \quad \frac{\partial u}{\partial z} = \frac{\partial^2 u}{\partial z^2} = 0
\]

the resulting coefficients are:
Boundary layer model

\[ a = 2 + \frac{n}{6}, \quad b = \frac{n}{2}, \quad c = -2 + \frac{n}{2}, \quad d = 1 - \frac{n}{6} \]  

Eq. 3.36

where

\[ \lambda = \frac{\delta^2 dU_e}{u} = -\frac{dp}{dx} \frac{\delta}{\mu U_e} \]  

Eq. 3.37

The resulting expression for the velocity profile is now inserted into the expression for the momentum thickness, Eq. 3.12 and displacement thickness Eq. 3.11. Integrating with respect to \( \eta \) along the profile gives an equation for \( \theta \) and \( \delta^* \) expressed in \( \lambda \) which instead can be rewritten to only depend on the boundary layer thickness \( \delta \) by introducing Eq. 3.37.

The expressions obtained are

\[ \theta(\delta) = \frac{37}{315} \delta - \frac{271}{945} \delta^3 \frac{du_e}{\nu dx} - \frac{1837}{9072} \delta^5 \left( \frac{du_e}{dx} \right)^2 \]  

Eq. 3.38

\[ \delta^*(\delta) = \frac{3}{10} \delta - \frac{41}{120} \delta^3 \frac{du_e}{\nu dx} \]  

Eq. 3.39

\[ \frac{\tau_{ee}}{\rho u_e^2} = \nu \left( 2 + \frac{\lambda}{6} \right) \]  

Eq. 3.40

By use of the chain rule Eq. 3.38 can be differentiated with respect to \( x \) and inserting Eq. 3.38-Eq. 3.40 into Eq. 3.19 with \( H = \frac{\delta^*}{\theta} \) gives a differential equation for \( \delta \) with respect to \( x \).

\[ \frac{d\theta}{d\delta} \frac{d\delta}{dx} + \frac{\theta(\delta)}{u_e} \frac{du_e}{dx} \left( \frac{\delta^*(\delta)}{\theta(\delta)} + 2 \right) = \frac{\nu}{u_e} \left( 2 + \frac{\delta^2}{6} \frac{du_e}{dx} \right) \]  

Eq. 3.41

This equation is then solved to obtain the boundary layer thickness and from Eq. 3.39 the displacement thickness is calculated. There exists no proper starting condition for Pohlhausen’s method so therefore the behaviour of the differential equation when \( x \) approaches zero needs to be studied. According to Schlichting (Reference 8) the laminar boundary layer at small \( x \) is proportional against \( \sqrt{x} \) for thin airfoils. By inserting a solution \( \delta = A \cdot \sqrt{x} \) (for \( x \) near 0) into the differential equation Eq. 3.41 and neglecting the terms that approach zero faster than \( \delta \) it is possible to obtain an equation from which the constant \( A \) is to be determined. This gives

\[ A = 2 \cdot \frac{315 \nu}{37 u_e} \]  

Eq. 3.42

Eq. 3.42 provides a starting condition for \( \delta \). The differential equation for all \( x \)-coordinates along the airfoil can then be solved.
3.3 Turbulent Solution Method

3.3.1 Background

Many calculations methods of turbulent flow over arbitrary bodies use the boundary layer equations along with some model of the turbulence. All these methods have their own advantages and disadvantages. Some are simple while some are more or less complicated to use. It is also possible to use the integral methods for laminar boundary layers with a logarithmic velocity profile and then calculate properties of a turbulent boundary layer, but this also leads to a cumbersome procedure. However, there is one method developed by Head that is very simple and straightforward that is one reason for looking closer at that method in this work.

3.3.2 Head’s Method for Turbulent Flow

This method is based on the concept of an entrainment velocity, which is defined as the rate of which the volumetric flow rate within the boundary layer increases in the x direction.

![Figure 11: Concept of Entrainment velocity](image)

With respect to Figure 11 the volumetric flow rate within the boundary layer is given by

\[ Q(x) = \int_{0}^{\delta(x)} u dy \]  
\[ \text{Eq. 3.43} \]

then the entrainment velocity is defined as

\[ E = \frac{dQ}{dx} \]  
\[ \text{Eq. 3.44} \]

The displacement thickness is written
\[
\delta^* = \int_0^\delta \left( 1 - \frac{u}{u_e} \right) dy = \int_0^\delta dy - \int_0^\delta \frac{u}{u_e} dy = \delta - \frac{Q}{u_e}
\]

Eq. 3.45

This gives

\[
Q = u_e (\delta - \delta^*)
\]

The entrainment velocity becomes

\[
E = \frac{dQ}{dx} = \frac{d}{dx} (u_e \theta H_1)
\]

Eq. 3.46

where

\[
H_1 = \frac{\delta - \delta^*}{\theta}
\]

Eq. 3.47

Now Head made the assumption that \(E/u_e\) only depends on \(H_1\) and by adapting experimental data Head came up with an equation for \(H_1\) given as

\[
\frac{1}{u_e} \frac{d}{dx} (u_e \theta H_1) = 0.0306 (H_1 - 3)^{-0.6169}
\]

Eq. 3.48

Now there are two equations for solving the flow inside the turbulent boundary layer, first the momentum integral Eq. 3.19 and second Eq. 3.48 above. These are two coupled differential equations that have to be solved simultaneously. But to be able to do this, numerical expressions for the friction coefficient \(C_f\) (used to determine the shear stress \(\tau_w\) at the wall) and \(H_1\) are needed.

Thus, Heads method consists of solving the following four equations

\[
\frac{d\theta}{dx} + \frac{1}{u_e} \frac{du_e}{dx} \theta (H + 2) = \frac{\tau_w}{\rho u_e^2}
\]

\[
\frac{1}{u_e} \frac{d}{dx} (u_e \theta H_1) = 0.0306 (H_1 - 3)^{-0.6169}
\]

\[H_1 = 3.3 + 0.8234 (H - 1.1)^{-1.287}, \quad \text{if } H \leq 1.6\]

\[= 3.3 + 1.5501 (H - 0.6778)^{-3.064}, \quad \text{if } H > 1.6\]

Eq. 3.49

\[
C_f = 0.246 \cdot 10^{-0.678\theta} \cdot Re^{-0.268}
\]

Eq. 3.50

These equations are solved for \(\theta, H_1\) and \(H\), it is then possible to compute the displacement thickness \(\delta^*\). To solve these equation two starting conditions, one for \(\theta\) and one for \(H\), is needed but this will be discussed later.
3.3.3 Prediction of Transition and Separation

So far two methods for predicting laminar and one for turbulent boundary layer growth have been presented. Thwaites and Head’s method require that the free stream distribution velocity to be known to start with while Pohlhausen assumes a velocity profile. By assuming a suitable starting condition for $\theta$ it possible to begin our calculations from the leading edge or a stagnation point with either Thwaites or Pohlhausen’s method where the flow is laminar. The calculation continues until the flow undergoes transition to turbulent flow. Whether the flow is turbulent or not can be determined by use of Michel’s formula which assumes transition when

$$\text{Re}_\theta > 1.174 \left( 1 + \frac{22,400}{\text{Re}_x} \right) \text{Re}_x^{0.46}$$  \hspace{1cm} \text{Eq. 3.51}$$

where the two Reynolds numbers can be defined as

$$\text{Re}_\theta = \frac{u_e \theta}{v} \quad \text{and} \quad \text{Re}_x = \frac{u_e x}{v}$$  \hspace{1cm} \text{Eq. 3.52}$$

From the point where transition occurs onwards Head’s method is employed. Starting condition for $\theta$ at this part is the value from the laminar model at the transition point. Head’s method also requires a starting value for the shape factor. Since this factor changes significantly during transition, one should instead of the laminar value use an empirical starting value for the turbulent boundary layer typical between 1.3 and 1.4. Sometimes laminar separation occurs before the flow undergoes transition. Separation for laminar conditions is indicated by $\lambda < -0.09$ or $H > 3.55$. Separation can also occur at turbulent conditions indicated by $H > 2.4$.

3.4 Solution Procedure of the Boundary Layer Problem

The solution procedure for the boundary layer problem will follow some basic steps listed below:

1. First an inviscid flow calculation for a given body shape is carried out (each wing station) and $u_e$ at the body surface is obtained.

2. Solve the boundary layer equations by means of any method described earlier and evaluate the displacement thickness $\delta^*$ in all calculation points.

3. From the obtained displacement thickness $\delta^*$ a new effective geometry is computed.

4. Repeat steps 1-3 until the obtained velocity $u_e$ do not deviate from the one obtained in the previous iteration.
A proper flow velocity around the body is achieved by the procedure above. It can now be used for calculating proper data along the body, such as lift coefficient curves etc.

3.5 Data Management

The implementation of the boundary layer model was quite straightforward but some modification in the model was necessary in order to make the program work properly for every type of body. The details of these approximations and modifications will be explained below.

3.5.1 Program Structure

The boundary layer program consists of six Matlab files/functions which are listed below:

1. blayer.m
2. blayersol.m
3. thwaite.m
4. head.m
5. headsolve.m

(1) blayer.m is the main program which requires the coordinates and Cp distribution around the current profile. It transforms the input coordinates so that the x axis is aligned along the body surface and \( x = 0 \) is situated in the stagnation point.

(2) blayersol.m determines if the boundary layer is laminar or turbulent.

(3) thwaite.m computes the displacement thickness as long as the boundary layer is laminar.

(4) head.m computes the displacement thickness as long as the boundary layer is turbulent with the coupled differential equations stored in (5) headsolve.m.

When the displacement thickness have been evaluated for both the lower and the upper surfaces it is added to the initial profile geometry and the new profile coordinates are obtained as output from the boundary layer program.

3.5.2 Method Modifications

Because of the simplicity in the calculation methods some problems where discovered during the implementation. As explained in section 3.2 and 3.3 these calculations are limited either to laminar or turbulent flows. But when the angle of attack increases separation may occur. In these regions neither Thwaites nor Pohlhausen’s methods are valid. Therefore the displacement thickness is set to be constant between the laminar separation point and the transition point, if not the flow reattaches and continues to be laminar. The same procedure is used in the turbulent area when Head’s method predicts turbulent separation. To avoid irregularities in the
solution linear interpolation was used from the point where turbulent separation occurs until the trailing edge of the airfoil is reached.

3.6 Summary of Computation Results Using the Boundary Layer Model

Thwaites and Pohlhausen’s methods were compared in validations, see [Appendix B]. In the resulting boundary layer model Thwaites method as used and the reason for this is that Pohlhausen’s method proved to be limited to very thin profiles. Pohlhausen’s method only gives accurate results for profiles with thickness ratio \( t/c \) less than 5%. When applied to thicker profiles the method gives completely unacceptable results. The problem with Pohlhausen’s method is to find a correct start value at the stagnation point and this is due to the sensitivity in the differential equation that is solved. If the start value differs only slightly the method gives an unreasonable result often with oscillations as a consequence. Thwaites method is much more robust for variations in start values and it gives reasonable results both for thin and thick profiles.

To be able to validate the resulting boundary layer model calculations around a flat plate at zero incidence. There exist analytical solutions for this case. In the second validation example pressure distributions around different airfoils were computed. Both with before and with the displacement thickness added. These were then compared with experimental results from wind tunnel testing.

In the validations above it was observed that the boundary layer model gives good results up to 15-20 degrees angle of attack. At higher angles most of the flow over the airfoil is separated and then the model is not valid anymore. Some of the resulting calculations of displacement and the boundary layer thickness can be seen in Figure 19-24. Illustrations of the resulting displacement thickness added to some airfoil geometries can be seen in [Appendix E].

In Figure 26 the resulting \( C_p \) distribution after more than one iteration with the boundary layer model can be seen. The results from the first iteration approach the experimental solution while the second and the third iteration start to diverge from the experimental solution. Therefore one iteration is used in the final boundary layer program. The boundary layer computation takes around 2 seconds for every wing station and it is not unusual to use up to 20-30 stations for each wing, especially when there exists flaps and ailerons as well. This would give a total computation time of 1-1.5 min for the entire aircraft.

3.7 Discussion of Boundary Layer Model

Analytical solutions and experimental data from wind tunnel tests were used to validate the boundary layer model. Compared to these the model gives excellent results. But as almost every integral method it shows problems when separation occur, therefore some modifications in the model were performed. These modifications will of course affect the result and the obtained values can be questionable when there are large areas of separated flow.

The two dimensional boundary layer integral methods used in this program has some advantages compared to other three dimensional methods. They are quite easy to implement, are computationally very fast, and they can be applied to any three dimensional configuration.
Most of these methods are quite accurate as long as the flow is primarily two dimensional (i.e., no significant cross flow exists), these methods are expected to break down in regions of large cross flow such as flow near separation, where most methods have difficulty. Remember that the intention with this work was to develop a simple boundary layer model which purpose is to give a preliminary estimate of boundary layer properties over a general configuration. From the validations performed it can be seen that the program gives quite good results for moderate angles of attack.

By use of interpolation computation time can be saved. The boundary layer displacement thickness is computed at e.g. every fourth station and then interpolation is used to find values at the stations left. This is an approximation but since the boundary layer method already is simple this is acceptable. Especially at high angles of attack, here the boundary layer model is known to give less reliable results. Performing calculations for every wing section might result in a very irregular wing which would imply in strange pressure distributions from the panel program.
CONCLUSIONS

A model for computing the cross flow around slender bodies have been developed together with a two dimensional boundary layer method which computes viscous effects around the aircraft wing.

Computation of flow properties around aircrafts with simple geometries can be done with the body model and a preliminary estimate of the viscous effects over the lifting surfaces with the boundary layer model. Despite that the method is rough it can be considered good enough for use in early design phases. Both methods have problems when the angle of attack gets high because of large areas with separated flow but here the use of potential theory fails and therefore the use of a more advanced computation program have to be considered. This is nothing unique for this program, most panel programs have this problem, and it is due to the approximations and simplifications done.

The body model has a similar problem when having bodies with high slenderness ratio and in this case it might be better to use the flat body description as an alternative, but this has to be tested out in future calculations with the models all connected to the original program (See Reference 1-2).

The methods developed are validated by performing flow calculations around both two dimensional and three dimensional bodies (body model) and wing profiles (boundary layer model). The results are good and differs less than a few percent from corresponding analytic solution. Both models use a low computation time and are suitable for use in early design phases where other more advanced calculation methods will be time consuming. The can be use a very simple geometry as input but the better the geometry used for describing the aircraft the higher accuracy in the results.
5 FUTURE WORK

Below some important improvements of the program developed in this thesis. The body model used here needs some more features to be included into the original program. The flow field around the aircraft will not be realistic in the ends of the reference box and a method for modelling the flow after the aircraft has to be introduced. This was beyond the scope of this work but one way to solve this is to place a wake behind the aircraft which reaches from the rear end towards infinity. This wake is then panelled in the same way as the outer reference box and a smooth transition at the ends of the reference box will be obtained. Else a sting such as used in wind tunnels can be used.

The body model also have problems at the ends of blunt bodies which can be solved by implementing some kind of analytical solution that is used in the cross sections closest to the nose and the rear of the aircraft. One such analytical solution that could work is the conformal mapping technique used in validation 1 [Appendix A].

Because of the approximations done in the separated areas the boundary model do not provide trustable results at high angles of attack. If the program is going to be used in this part of the flight envelope a better modelling of the displacement thickness is necessary. The calculation time on the other hand may not be affected because only a slight increase, say 2 seconds/station will double the total time needed for the boundary layer contribution.

The purpose with the different validations was to find the models limitations and how many panels needed to give results with enough accuracy, but it is still quite simple configurations that have been used and before the program becomes operative there should be some more validations performed. For example it would be useful to compare results from these methods with results from calculations with the similar NASA wing-body program, results from CFD calculations or experimental data from wind tunnel tests, this concerns both the body and the boundary layer model.
6 REFERENCES


11. Personal communications with Per Weinerfelt, Department of Aerodynamics & Flight mechanics, Saab Aerosystems.

12. NASA Technical report server  
   [http://techreports.larc.nasa.gov/]  2005-10-21

13. American Institute of Aeronautics and Astronautics  

14. Document with advanced boundary layer theory  

15. A boundary layer program  
   [http://www.genie.uottawa.ca/~mcg4345/boundarylayer.m]  2005-12-01
APPENDICES

Appendix A: Validation of Body Contribution

In order to validate the body contribution some different problems for which it is possible to obtain an analytical solution exist were used. First the accuracy of the two dimensional solver was controlled, then calculations around three dimensional bodies were validated.

Validation 1: 2D Flow Over a Wing Body Configuration.

The two dimensional cross flow for a lifting slender body can be obtained by using conformal transformation. This solution can be found in Ashley & Landahl (Reference 3) and a picture of the corresponding geometry can be seen in Figure 12. This method combines the results from two classical flow situations. Flow around a circular cylinder is applied to the equivalent body of revolution and the flow over a flat plate normal to the stream is used for modelling flow around a thin slender wing. This is of course an approximation but it is a good one. A practical example that incorporates these solutions is that of a mid-winged body of revolution as in the picture below.

This type of solution is only valid for slender bodies with lifting or axial flow, here the axial flow will keep the cross flow from separate when it passes the body. Note that if the angle of attack becomes to large then the flow separates, and the solution is not valid anymore. The analytical method used here also uses conformal mapping to transform the body into more suitable geometries.

![Figure 12: Wing - Slender body combination](image)

From the discussion above the problem becomes that of finding a two dimensional flow having a non dimensional vertical velocity of

\[ w_{\alpha} = \frac{W_{\alpha}}{U_{\infty}} = \sin \alpha \approx \alpha \]  

Eq. 7.1
The solution for this type of configuration can in short terms be derived from the corresponding expression for the velocity potential

\[ \varphi' = \alpha \cdot z \quad \text{Eq. 7.2} \]

By use of a complex potential \( \varphi' \) can be obtained from

\[ w'(X) = \varphi'(y, z) + i \psi(y, z) \quad \text{where} \quad X = y + i \cdot z \quad \text{Eq. 7.3} \]

The complex potential can now be constructed by use of two conformal transformations, of which the first is called the Joukowsky transformation

\[ X_1 = X + \frac{R^2}{X} \quad \text{Eq. 7.4} \]

Which conforms the outside contour onto a slit along the \( y_1 \)-axis of width \( 2s_1 \) and

\[ s_1 = s + \frac{R^2}{s} \quad \text{Eq. 7.5} \]

\[ X_2 = \left( X_1 - s_1 \right)^{1/2} \quad \text{Eq. 7.6} \]

The second transforms the horizontal slit to a vertical slit of width \( 2s_1 \) which then corresponds to a flat plate with zero angle of attack compared with the cross flow. The flow in the \( X_2 \) plane can now be written

\[ W'(X_2) = -i \alpha X_2 \quad \text{Eq. 7.7} \]

Now substituting Eq. 7.4 and Eq. 7.6 into Eq. 7.7 gives

\[ w'(X) = -i \alpha \left[ \left( X + \frac{R^2}{X} \right)^2 - \left( s + \frac{R^2}{s} \right)^2 \right]^{1/2} \quad \text{Eq. 7.8} \]

The complex velocity perturbation potential can now be obtained by adding the free stream potential

\[ W(X) = W'(X) + i \alpha X = -i \alpha \left[ \left( X + \frac{R^2}{X} \right)^2 - \left( s + \frac{R^2}{s} \right)^2 \right]^{1/2} - X \quad \text{Eq. 7.9} \]
With \( X \) from Eq. 7.3 and after some simplification an equation that has one real part and one imaginary are obtained. The real is the velocity potential \( \varphi \) and the imaginary is the stream function \( \psi \). An easy manner to compare these solutions is to calculate the velocity potential \( \Phi \) in a net of grid points inside the two dimensional flow region and by using Matlab it is possible to make graphs with contour lines which then can be compared with the analytic solution. To validate the program, calculations with different flow speeds and angles of attack were performed. Here are only two different results are shown, one with a circular cross section (Figure 13-14) and a circular body with low aspect ratio mid wings (Figure 15-16). Both are exposed to a lifting flow \( U_x = [100, 0.5] \) m/s which corresponds to an angle of attack, \( \alpha \approx 3^\circ \). The theoretical solutions and the panel program uses \( \phi = 1 \) as boundary condition along the outer boundary.

![Contour plot - Theoretical solution](image-url)

**Figure 13: Contour graph- Theoretical solution**
As seen in the figures the analytical solution and the panel method gives almost identical results. In Figure 15 and Figure 16 there is an excellent agreement between the theoretical solution and the analytical. Note that there are some disturbances in the region close to the wings in both the analytical and panel program solution. These depend of how many panels and grid points used. When the number of grid points and panels along the boundaries are increased these disturbances gets much smaller.
Figure 15: Contour graph – Wing Body - Theoretical solution
(Observe that the wings are missing in the figure but they are included in the calculation)

Figure 16: Contour graph - Wing Body - Panel method
Validation 2: 3D Axial Flow Over a Parabolic and an Elliptic Shaped Body.

In the next validation case the flow field over two different three dimensional bodies of revolution was computed. The bodies have the same length and maximal thickness/diameter but one of them is parabolic and the other is elliptic shaped. In this case the pressure coefficient $C_p$ was used to compare the analytical solution with the one obtained from the panel program. Two different analytical solutions were used when comparing the flow over the two bodies; the first analytical solution is derived directly from the integral functions used in the panel method (Eq. 2.20-Eq. 2.21 in the theory part without the discrete summation) and will not be described further. See (Reference 11). The second one is taken directly from Schlichting (Reference 5) that has derived a formula for the disturbance velocity over both a parabolic and an elliptic shaped body. From the velocity the pressure distribution can be derived using Eq. 2.40 in the theory.

From Schlichting (Reference 5) the following expressions for the perturbation velocity along the bodies are taken.

Parabolic:

$$\frac{u(X)}{U_\infty} = 2[1 - 6X(1 - X)][3 + \ln X(1 - X) + 2 \ln \delta_R] \cdot \delta_R^2$$

Eq. 7.10

Elliptic:

$$\frac{u(X)}{U_\infty} = -\left[\frac{1}{4X(1 - X)} + \ln \left(\frac{\delta_R}{\delta_R} \right)\right] \cdot \delta_R^2$$

Eq. 7.11

where $\delta_R$ is the maximal body diameter.

In the calculations the body length $l_1 = 1$ was used. The potential $\phi$ at the outer reference box used during this validation was obtained by another formula from Schlichting, hence

$$\phi = -\frac{1}{4\pi} \int_0^{l_1} \frac{q(x') dx'}{\sqrt{(x-x')^2 + r^2}}$$

Eq. 7.12

The function $q(x')$ looks like

$$q(x') = \pi \cdot U_\infty \cdot \frac{d(R^2)}{dx}$$

Eq. 7.13
Notice that Eq. 7.13 is only dependent of the body shape since the only non constant variable is the body radius $R$.
This formula is then integrated over the body’s length and with $r$ corresponding to all control points on the outer boundary. This gives the potential $\phi$ which then is used as boundary condition when evaluating the two dimensional cross flow.

The resulting $C_p$ curves can be seen in Figure 17-18. As seen in the figures the accuracy for the parabolic body is very good while the elliptic differ a little bit in the nose region and at the tail. These differences depend on the derivatives with respect to the normal vector which tends to infinity at the ends of the body.

The calculations below are performed with 40 panels on the outer boundary, 20 on the inner and the bodies are divided into 100 sections).
As can be seen in the figures above the accuracy are good despite the fact that the number of panels and sections used is low. The calculation time for these configurations is between 10 and 20 seconds/body. Even when performing a few more iterations the results converge quickly and the calculation time will be low.

Several calculations were made on these bodies to find out more exactly how many panels and sections that have to be used to get a good enough agreement between the panel method and the analytical solution. Because of the fact that the equations used to solve the problem are only valid for more or less slender bodies there are calculations around bodies with different slenderness ratio performed. This could give knowledge about which type of bodies where it is possible to use the program and still get a result with enough accuracy. At last also the size and the shape of the outer box were changed to find the optimal combination between the current body size/shape and the outer boundary.

The different solutions were compared by using the maximum error between the different curves, which is obtained from

$$\text{max}_{err} = \frac{C_{p,\text{panel}} - C_{p,\text{analytical}}}{C_{p,\text{analytical}}} \cdot 100$$

Eq. 7.14

In the tables below there is a summary of the different tests and the maximal error for each configuration. The error below is measured between the analytical solution from Schlichting and the result obtained from the panel method.
Parabolic body

Refinement in the x-direction

Outer boundary: 40 panels/section
Inner boundary: 20 panels/section
Outer boundary box size: $1.4 \cdot r_{\text{body-max}}$

<table>
<thead>
<tr>
<th>Nr. sections</th>
<th>Max. error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>50</td>
<td>11.51</td>
</tr>
<tr>
<td>80</td>
<td>9.61</td>
</tr>
<tr>
<td>100</td>
<td>0.46</td>
</tr>
<tr>
<td>150</td>
<td>0.15</td>
</tr>
<tr>
<td>200</td>
<td>0.8</td>
</tr>
<tr>
<td>300</td>
<td>2.47</td>
</tr>
</tbody>
</table>

Refinement of nr panels at the body

Number of sections: 100
Outer boundary: 40 panels/section
Outer boundary box size: $1.4 \cdot r_{\text{body-max}}$

<table>
<thead>
<tr>
<th>Nr. panels</th>
<th>Max. error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>1.26</td>
</tr>
<tr>
<td>15</td>
<td>0.66</td>
</tr>
<tr>
<td>20</td>
<td>0.46</td>
</tr>
<tr>
<td>30</td>
<td>0.0027</td>
</tr>
<tr>
<td>40</td>
<td>0.21</td>
</tr>
<tr>
<td>50</td>
<td>0.18</td>
</tr>
</tbody>
</table>

Refinement of nr panels at the outer boundary

Number of sections: 100
Inner boundary: 20 panels/section
Outer boundary box size: $1.4 \cdot r_{\text{body-max}}$

<table>
<thead>
<tr>
<th>Nr. panels</th>
<th>Max. error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>0.626</td>
</tr>
<tr>
<td>20</td>
<td>0.46</td>
</tr>
<tr>
<td>30</td>
<td>0.39</td>
</tr>
<tr>
<td>40</td>
<td>0.37</td>
</tr>
<tr>
<td>50</td>
<td>0.37</td>
</tr>
</tbody>
</table>

Variation of the outer box size:

Number of sections: 100
Outer boundary: 40 panels/section
Inner boundary: 20 panels/section

<table>
<thead>
<tr>
<th>Size factor</th>
<th>Max. error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$1.2 \cdot r_{\text{in-max}}$</td>
<td>2.39</td>
</tr>
<tr>
<td>1.4</td>
<td>0.46</td>
</tr>
<tr>
<td>1.6</td>
<td>1.31</td>
</tr>
<tr>
<td>1.8</td>
<td>3.02</td>
</tr>
<tr>
<td>2</td>
<td>4.72</td>
</tr>
<tr>
<td>2.2</td>
<td>4.83</td>
</tr>
</tbody>
</table>

Change of outer box to cylinder and variation of the number of panels.

Number of sections: 100
Inner boundary: 20 panels/section
Outer boundary box size: $1.4 \cdot r_{\text{body-max}}$

<table>
<thead>
<tr>
<th>Nr. panels</th>
<th>Max. error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>0.626</td>
</tr>
<tr>
<td>20</td>
<td>0.426</td>
</tr>
<tr>
<td>30</td>
<td>0.39</td>
</tr>
<tr>
<td>40</td>
<td>0.378</td>
</tr>
<tr>
<td>50</td>
<td>0.36</td>
</tr>
</tbody>
</table>

Variation of the slenderness ratio of the body

$R_{body} = amp \cdot x(1-x)$

Number of sections: 100
Inner boundary: 20 panels/section
Outer boundary: 40 panels/section
Outer boundary box size: $1.4 \cdot r_{\text{body-max}}$

<table>
<thead>
<tr>
<th>Body thickness (amp)</th>
<th>Max. error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.05</td>
<td>0.37</td>
</tr>
<tr>
<td>0.1</td>
<td>0.39</td>
</tr>
<tr>
<td>0.2</td>
<td>0.46</td>
</tr>
<tr>
<td>0.4</td>
<td>24.54</td>
</tr>
<tr>
<td>0.6</td>
<td>85</td>
</tr>
<tr>
<td>1</td>
<td>-</td>
</tr>
</tbody>
</table>
The results from the validations with the elliptic body are similar to the one above. From the tables above it is possible to make some conclusions. The accuracy gets better the more sections and panels used at the boundaries. The user is therefore recommended to use somewhere between 100 - 150 sections along the body, 20-30 panels/section at the body surface and 20-40 panels/section at the outer boundary. This gives both results with good accuracy and with low calculation time for bodies with simple fuselage/ engine geometries. When it comes to the outer box size there is no hesitation about that \( 1.4 \cdot r_{\text{body-max}} \) is the size which gives the best accuracy which also is the validation with the elliptic body verifies. Notice also that when changing the reference box into a cylinder there is no difference in the results compared to the rectangular shape.

The body thickness/size is interesting, as seen in the table the difference is quite large already at \( \text{amp} = 0.4 \) which corresponds to a body of maximum diameter 0.2 and that is a slenderness ratio of 1/5. This might be misleading because when using Schlichting’s equations to compute the boundary conditions an assumption valid only for slender bodies is made. This is an approximation made to achieve boundary condition needed to calculate the flow around the body with the panel program which then affects the results.

**Appendix B: Validation of Boundary Layer Model**

**Validation 1: Flow Over a Flat Plate with Zero Incidence**

To validate the boundary layer methods the flow over a flat plate with zero incidence is used because it has a simple analytical formulas.

The solution of the boundary layer equations for a flat plate is called the Blasius solution and it leads to the following relations for the boundary layer parameters.

**Laminar flow (starting from the leading edge)**

\[
\delta = 5 \cdot x / \sqrt{\text{Re}_x} \quad \delta^* = 1.721 \cdot x / \sqrt{\text{Re}_x} \quad \text{where} \quad \text{Re}_x = \frac{\text{u}_x \cdot x}{\nu}
\]

**Turbulent flow (starting from the leading edge)**

For turbulent flow the relations changes with Reynolds number but according to Young (Reference 7) a good approximation for the boundary layer thickness is

\[
\delta = 0.37 \cdot x / (\text{Re}_x)^{1/5} \quad \text{for Reynolds numbers around} \ 5 \cdot 10^5 \ (\text{transition}) \ to \ 10^7.
\]

For higher Reynolds numbers a better approximation is

\[
\delta = 0.27 \cdot x / (\text{Re}_x)^{1/6}
\]

The displacement and the momentum thickness also varies slightly with Reynolds number but can be approximated with the relations

\[
\delta^* = 0.018 \cdot x / (\text{Re}_x)^{1/7} \quad \theta = 0.0142 \cdot x / (\text{Re}_x)^{1/7}
\]
Validation of Thwaites method-Pohlhausen’s method
In the first test case a low speed flow was considered so that transition does not occur. In Figure 19-20 the boundary layer thickness and the displacement thickness along the plate with $x = 0...1$ and flow speed $U_\infty = 10 \text{ m/s}$ can be seen.

Validation of Head’s method
By increasing the flow speed transition occurs close to the leading edge. In Figure 21-22 the boundary layer and the displacement thickness along the plate can be seen for Reynolds numbers between $5 \cdot 10^5 - 10^7$. Figure 23 shows the boundary layer thickness for Reynolds numbers above $10^7$ (BL = boundary layer).

![Graph showing boundary layer thickness](image)

**Figure 19:** BL thickness for a flat plate at laminar conditions
(Thwaites method, Pohlhausen’s method)
As can be seen in the figures above both Thwaite and Pohlhausen’s methods for predicting laminar boundary layer give results that agree quite well with the analytical for a flat plate.

Figure 21: BL thickness for a flat plate at turbulent conditions, Reynolds numbers less than $10^5$

(Head’s method)
Figure 22: Displacement thickness for a flat plate with turbulent BL
(Head’s method)

Figure 23: BL thickness for a flat plate at turbulent conditions, Reynolds numbers higher than $10^7$
(Head’s method)
Head’s method corresponds to the analytical solutions but here are the differences between the curves a little bit larger. Observe that there still is a small piece in the beginning of the boundary layer which is laminar and this will contribute to the error. For the higher Reynolds number case the error between Head and the analytical is smaller and there’s also a shorter laminar boundary layer in the beginning.

**Validation 2: Prediction of Boundary Layer Over Airfoils**

This validation is performed by comparing computed pressure distribution over an airfoil (including the displacement thickness added to the profile geometry) with experimental data obtained from wind tunnel tests. The airfoil used is a modified NACA LS(1)-0417. The geometry of this airfoil can be seen in Figure 24. Some of the obtained cp distributions are shown in Figure 25-26.

The difference between the different curves is small but it is possible to see that the solution with the boundary layer contribution approaches the experimental solution. The strange behaviour at the trailing edge is created when the cp distribution is computed. Instead of using the entire panel program (which would have taken a lot more work) an already existing 2D vortex panel code developed at NASA was used when performing this validation. The problem occurs when the thickness of the airfoil increases due to boundary layer there is need of using panels also along the trailing edge but that is not a feature in this panel code.

In Figure 26 the pressure distribution after two iterations with the boundary layer model are showed. The second and the third iterations differs significantly more than the one obtained from the first iteration and this behaviour is the same for most of the different types of airfoils tested in this validation. Therefore it becomes evident to use one iteration in the final boundary model.

![Figure 24: Geometry of NACA LS(1)-0417 mod., boundary layer at 8 degree angle of attack.](image)

Re=2\times10^6
- Appendices -

Figure 25: Cp distribution around NACA LS(1)-0417 mod. airfoil at 8 degree angle of attack.
\[ \text{Re}=2\times10^6 \]

Figure 26: Cp distribution around NACA 0417 mod., 2 iterations
\[ \text{Re}=2\times10^6 \]
Appendix C: Derivation of Integral Coefficients

In this appendix the integral coefficient used in the linear equation system describing the potential Eq. 2.32 and the velocity Eq. 2.33 in the inner area are derived. See section 2.3.1 for the theory behind these equations.

Derivation of the integral coefficients used in the velocity potential calculation in the inner area

In Eq. 2.30 and Eq. 2.31 $\phi_{o,j}$ at the outer boundary is known. It can be obtained outer problem and at the first iteration $\phi_{o,j}=0$ as start solution. These equations can be rewritten so that the constant terms form the right hand side of the equation.

\[
\sum_{i=1}^{N} \frac{\phi_{o,j}}{2\pi} \int_{0}^{\theta} \rho \left( \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) \left[ \ln|\rho| \right] dp - \sum_{i=1}^{M} \frac{1}{\phi_{o,n}} \int_{0}^{\theta} \rho \left( \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) \left[ \ln|\rho| \right] dp = \]
\[\left( A_{o}, \theta \right) \]
\[
\phi_{o,j}(y_{j},z_{j}) - \sum_{i=1}^{M} \frac{\phi_{o,n}}{2\pi} \int_{0}^{\theta} \rho \left( \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) \left[ \ln|\rho| \right] dp + \sum_{i=1}^{N} \frac{1}{\phi_{o,n}} \int_{0}^{\theta} \rho \left( \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) \left[ \ln|\rho| \right] dp \]
\[\left( A_{o}, \theta \right) \]
\[
\sum_{i=1}^{N} \frac{1}{2\pi} \int_{0}^{\theta} \rho \left( \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) \left[ \ln|\rho| \right] dp = \sum_{i=1}^{M} \frac{\phi_{o,n}}{2\pi} \int_{0}^{\theta} \rho \left( \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) \left[ \ln|\rho| \right] dp \]
\[
\left( A_{o}, \theta \right) \]

In equations Eq. 7.15-7.16 the integrals are computed exactly using Matlab. They are simplified as much as possible before performing the integration. Rewrite the equations according to:

\[\int_{0}^{\theta} \ln(|\rho|) dp = \int_{0}^{\theta} \ln \left( (v-y_{i})^2 + (z-z_{i})^2 \right) dp = \frac{1}{2} \int_{0}^{\theta} \ln(ap^2 + 2bp + c) dp \]

In the cases where $\rho \left( \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right)$ occurs, the logarithm above is differentiated with respect to $y$ and $z$ coordinates.

The following expressions are obtained
\[ \pi l \left( \frac{\partial}{\partial y} \cdot \frac{\partial}{\partial z} \right) \ln \left( \sqrt{(y-y_1)^2 + (z-z_1)^2} \right) = \left( \frac{y-y_1}{\sqrt{(y-y_1)^2 + (z-z_1)^2}} - \frac{z-z_1}{\sqrt{(y-y_1)^2 + (z-z_1)^2}} \right) \cdot (\Delta z, -\Delta y) = \]  

Eq. 7.18

Inserting expressions for \( y \) and \( z \) from Eq. 2.24, doing simplifications the expression and with the denominator rewritten as a second degree polynomial and substituting the different expressions with the constants \( a, b, c \) and \( e \). Eq. 7.18 takes the form:

\[ \Rightarrow \frac{\Delta z \cdot (y-y_1) - \Delta y \cdot (z-z_1)}{\sqrt{(y-y_1)^2 + (z-z_1)^2}} dp = \frac{e}{ap^2 + 2bp + c} dp \]  

Eq. 7.19

where

\[ a = \Delta y^2 + \Delta z^2 \]  
\[ b = \Delta y (y_i - y_1) + \Delta z (z_i - z_1) \]  
\[ c = (y_i - y_1)^2 + (z_i - z_1)^2 \]  
\[ e = \Delta z (y_i - y_1) - \Delta y (z_i - z_1) \]  

Eq. 7.20

The integration of Eq. 7.19 can now easily be performed with Matlab resulting in

\[ \frac{1}{2\pi} \arctan \left( \frac{a+b}{e} \right) - \frac{1}{2\pi} \arctan \left( \frac{b}{e} \right) \]  

Eq. 7.21

All the integral coefficients computed and inserted into equations Eq. 7.15 and Eq. 7.16 lead to

Outer boundary (Eq. 7.15):

\[ (A_{21} \cdot \phi) + \left( A_{22} \cdot \left( \frac{\partial \phi}{\partial n} \right) \right) = B_{2}(j) \]  

Eq. 7.22

Inner boundary (Eq. 7.16):

\[ (A_{11} \cdot \phi) + \left( A_{12} \cdot \left( \frac{\partial \phi}{\partial n} \right) \right) = B_{1}(i) \]  

Eq. 7.23

Where \( j = 1-M \) (number of panels on the outer boundary) and \( i = 1-N \) (number of panels on the inner boundary)
Eq. 7.22-Eq. 7.23 forms a linear equation system describing the velocity potential in the inner solution area.

\[
\begin{pmatrix}
A_{11} & A_{12} \\
A_{21} & A_{22}
\end{pmatrix}
\begin{pmatrix}
\frac{\partial \phi}{\partial n} \\
\phi
\end{pmatrix}
= \begin{pmatrix}
B_1 \\
B_2
\end{pmatrix} \iff A \cdot \Phi = B
\]

Eq. 7.24

Derivation of the integral coefficients used in the velocity calculation in the inner area

When the potential \( \phi \) is known at the inner boundary the flow speed can be computed from the relation

\[ V = \nabla \phi \] 

Eq. 7.25

Differentiating Eq. 2.23 and replacing left hand side by Eq. 7.25 yield

\[
V(y_j, z_j) = \sum_{i=1}^{N} \frac{\phi_{i,j}}{2\pi} \left[ \frac{\partial}{\partial n_i} \ln \rho \right] dp - \sum_{i=1}^{N} \frac{1}{2\pi} \left( \frac{\partial \phi_{i,j}}{\partial n_i} \right) \left[ \ln \rho \right] dp + \sum_{i=1}^{N} \frac{\phi_{i,j}}{2\pi} \left[ \frac{\partial}{\partial n_j} \ln \rho \right] dp - \sum_{i=1}^{N} \frac{1}{2\pi} \left( \frac{\partial \phi_{i,j}}{\partial n_j} \right) \left[ \ln \rho \right] dp
\]

Eq. 7.26

Observe that the normal vector \( n_j \) is valid only in the control point that is evaluated. In the case where the second derivative occurs a rewriting that simplifies the integration is performed. For example, take the first integral coefficient in Eq. 7.26 can be rewritten as

\[
\frac{\partial}{\partial n_j} = n_j \left( -\frac{\partial}{\partial y_1}, -\frac{\partial}{\partial z_1} \right)
\]

Eq. 7.27

Where \( y_1 \) and \( z_1 \) is the control point coordinate.

First by taking the derivative with respect to \( y \)

\[
\frac{\phi_{i,j}}{2\pi} \left( -\frac{\partial}{\partial y_1} \right) \int \left[ \frac{\partial}{\partial n_i} \ln \rho \right] dp = \frac{\phi_{i,j}}{2\pi} \left( -\frac{\partial}{\partial y_1} \right) \int \left( \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) (dz, -dy) ds = \]

\[
\frac{\phi_{i,j}}{2\pi} \int \left( \frac{\partial^2}{\partial y^2}, \frac{\partial^2}{\partial z \partial y} \right) (dz, -dy) ds = \frac{\phi_{i,j}}{2\pi} \int \left( \frac{\partial^2}{\partial z^2}, -\frac{\partial^2}{\partial z \partial y} \right) (dz, -dy) ds
\]

Eq. 7.28

using \( \Delta |\delta| = 0 \) gives
Thus, the only term left is a finite difference which easily can be calculated. The same procedure holds for the derivative with respect to $z$

$$\frac{\phi_{i,j}}{2\pi} \left[ \frac{\partial \ln |\rho|}{\partial y} \right] \bigg|_{(y_{i},z_{i})}$$

Eq. 7.30

Together and multiplied with the normal vector in Eq. 7.27 gives

$$\frac{\phi_{i,j}}{2\pi} \left[ \frac{\partial \ln |\rho|}{\partial y} \right] \left[ \tilde{n}_{y,j}, \frac{\partial \ln |\rho|}{\partial z} - \frac{\partial \ln |\rho|}{\partial y} \right]_{j}^{z_{j}}$$

Eq. 7.31

The remaining integral coefficients in Eq. 7.26 are computed analytically using symbolic calculation in Matlab. Then the Eq. 7.26 is rewritten as an equation system in the same manner as the potential, thus

$$V_{i} = C_{11} \cdot \phi_{i} + C_{12} \cdot \left( \frac{\partial \phi}{\partial n} \right)_{i} + C_{21} \phi_{o} + C_{22} \left( \frac{\partial \phi}{\partial n} \right)_{o}$$

Eq. 7.32

Due to complexity the different expressions for the constants $C_{11}$, $C_{12}$ and $D_{1}$ are left out. Eq. 7.32 describes the disturbance velocity in the inner solution area.
Appendix D: Figures Body Model

Figure 27: Example aircraft 1, civil passenger aircraft

Figure 28: Example aircraft 1, civil passenger aircraft with outer reference box
Figure 29: Example aircraft 2, two engine aircraft with V-tail and reference boxes
(Note the three bodies of which this aircraft consists)

Figure 30: Example aircraft 3, UAV with reference box
Appendix E: Figures Boundary Layer Model

Figure 31: Boundary layer around NACA 4412 at 8 degree angle of attack
Re=2*10^6

Figure 32: Boundary layer around NACA 0010 at 5 degree angle of attack
Re=2*10^6
Figure 33: Displacement thickness - Upper surface NACA 0010, alpha=5 degrees.
$Re=1\times10^6$, $0 < x < 0.09$ Laminar BL, $0.09 < x < 1$ Turbulent BL.

Figure 34: Displacement thickness - Lower surface NACA 0010, alpha=5 degrees.
$Re=1\times10^6$, $0.01 < x < 1$ Turbulent BL.
Sammanfattning

The aim of this thesis was to develop a potential flow calculation model which includes computation of flow around aircraft bodies (fuselage, engines) and a boundary layer method which calculates the viscous effects over the aircraft wings. The models developed will be merged with of an already existing panel program developed at Saab, Linköping, Sweden. Different methods have been studied but the basis of the work have been to develop a model using a panel method which can provide results from a simple geometry description, with short calculation time and hence be used in early design phases. In this thesis Matlab have been used as programming language, this to ensure that future development and maintenance is possible.

The body model uses a panel method where the flow domain is divided into an inner and an outer part where the outer problem uses a three dimensional panel description while the inner problem performs two dimensional calculations. The inner and outer problems are separated by an arbitrarily shaped reference box. The inner area is divided into a number of cross sections which are described by line segments. With the help of these the two dimensional cross flow is obtained. This result is connected to the outer part trough boundary conditions and the entire three dimensional flow domain can be determined.

The resulting body program is limited to aircraft bodies with a slenderness ratio less than 1/5. Higher values violate the model assumption. The number of cross sections needed to describe a body of one unit length is between 80-150 and the number of line segments needed for one cross sections is 20 for the inner boundary and 40 line segments for the outer. This configuration gives results with acceptable accuracy within a computation time less than 15 seconds/body.

The viscous effects around the aircraft wings are modelled with a two dimensional boundary layer model where the boundary layer displacement thickness over the wing profile is calculated with two different methods depending on if the flow in the boundary layer is laminar or turbulent. The computed displacement thickness is then added to the wing profile geometry and new pressure distributions are computed on the modified geometry.

The computed pressure distributions including the viscous effects show better agreement with results from experimental wind tunnel tests than the inviscid without boundary layer contribution. Separation is not modelled and neither the large effects this has on the pressure distribution. The model gives useable results up to 15-20 degrees angle of attack; at higher angles the separated regions are so large that the model is not valid anyway.

The work was performed at the department of applied aerodynamics and flight mechanics at Saab Aerosystems during the period 2005-08-24 – 2006-01-31.