Graduation Project at the Queens University of Belfast

Experimental study of an incompressible laminar flow over a 3-D rectangular cavity

Supervisors Queens University: Prof. S. R. Raghunathan
Dr. R. K. Cooper
Dr. E. Benard

Supervisors Technische Universiteit Eindhoven: Prof. dr. ir. A. Hirschberg
Prof. dr. ir. M.E.H. v. Dongen

Jochim Vandebroek

Technische Universiteit Eindhoven
Acknowledgement

I would like to thank my supervisors Professor S. R. Raghunathan, Dr. Richard K. Cooper and Dr. Emmanuel Benard at the School of Aeronautical Engineering of the Queen’s University of Belfast. Their knowledge and insight have been very useful to me. Also the help of Dr. Andrew Sidorenko was very welcome to understand the operation of the experimental setup.

I am grateful to the people who worked at the School of Aeronautical Engineering and helped me with my study: Dr. Hui Yao for the numerical calculations of the cavity flow, Kafeel Ahmed, G.-K. Kerevanian and Dave Robinson for their time and support!

I am also very thankful to my supervisors at the Technische Universiteit Eindhoven, Professor dr. ir. A. Hirschberg and Professor dr. ir. M.E.H. v. Dongen, for their motivation during the writing of my report.

I would most like to thank my family and my girlfriend Veerle for their help and support during the time of this study!
## Content

1. **Summary**  
   
2. **Introduction**  
   
3. **Flow in and around cavities, a literature survey**  
   3.1 Introduction  
   3.2 Boundary layers  
   3.3 Boundary layer properties  
      3.3.1 Displacement thickness $\delta^*$  
      3.3.2 Momentum thickness $\theta$  
      3.3.3 Shape factor $H$  
   3.4 Turbulent boundary layer  
   3.5 The laminar-turbulent transition  
   3.6 Primary stability theory  
   3.7 Flow over a cavity  
   3.8 History review of cavity flow studies  
      3.8.1 Experimental studies  
      3.8.2 Computational studies  
      3.8.3 Topic of present study  
   3.9 Related research projects  
      3.9.1 Research on 2-D cavities  
      3.9.2 Research on 3-D cavities  
      3.9.3 Varying the width of the cavity  

4. **Hot-wire anemometry**  
   4.1 Introduction  
   4.2 Effective cooling velocity method  
   4.3 Probe calibration  
      4.3.1 The square wave test  
      4.3.2 SN-Probe calibration
5. Experimental setup

5.1 Introduction
5.2 The wind-tunnel
5.3 Flat plate model with the cavity
5.4 Transverse system
5.5 Hot-wire anemometer
5.6 Data acquisition

6. Experimental results

6.1 Introduction
6.2 Visualization methods
   6.2.1 Miniature balls
   6.2.2 Light reflecting liquid
6.3 Results outside the cavity
   6.3.1 Boundary layer profile for mean velocity
   6.3.2 Boundary layer thickness $\delta_{0.99}$
   6.3.3 Displacement thickness $\delta'$
   6.3.4 Momentum thickness $\theta$
   6.3.5 Shape factor H
   6.3.6 Pulsation profile

7. Numerical results

7.1 Computational studies
7.2 Comparison theoretical and experimental data

8. Conclusion

Appendix A: Equations for incompressible Newtonian fluids
Appendix B: The Blasius profile
Appendix C: Pohlhausen’s method for a laminar boundary layer
Appendix D: Von Karman momentum equation
Appendix E: Turbulent boundary layer theory
Appendix F: Derivation of the Orr-Sommerfeld equation
Appendix G: Matlab program to process data
Appendix H: Power spectra

References
1. Summary

Flow fields around cavities show unsteady velocity, density and pressure fluctuations. Landing-gear wells and gaps in the shell of an aircraft, for example, have a strong influence on the flow characteristics. These effects may lead to damaged materials or structural failure. In order to understand this complex flow structure, experimental studies were performed in a wind-tunnel on a 3-dimensional open cavity, assuming a rectangular geometry. This report describes the incompressible flow past an open shallow cavity with a depth-to-length ratio (D/L) of 0.25 and a laminar Blasius boundary layer as inflow condition. This study is also used to verify the laminar-to-turbulent transition prediction of the boundary layer cavity flow from numerical CFD results made by Hao at the Queen’s University of Belfast.

To study the flow field inside the cavity, the following visualization methods are used. Miniature balls are placed randomly at the bottom of the cavity and the movement of these balls is studied once the wind tunnel is switched on. Also a light reflecting liquid is spread out inside the cavity to obtain a picture from the flow pattern at the bottom of the cavity. To obtain information about the flow field outside the cavity, velocity measurements are taken at different positions. Therefore the method of single hot-wire anemometry is employed. A probe is placed inside the boundary layer and traversed step by step to obtain boundary layer velocity profiles. Based on these results, several boundary layer properties are deduced like the boundary layer thickness $\delta$, the displacement thickness $\delta^+$, the momentum thickness $\theta$ and the shape factor $H$. Also the pulsation velocity profiles and the pulsation intensities are examined. To conclude, a comparison between experimental and numerical values has been made: the numerical CFD results for the boundary layer profiles are compared with the experimental data.

The prediction of laminar-to-turbulent transition by the computational studies is confirmed by the experimental results. The laminar profiles upstream of the cavity become turbulent boundary layer profiles downstream of the cavity as shown by the parameters discussed above. The influences of the cavity corners and the re-circulating flow inside the cavity are of great importance for the flow around the cavity.
2. Introduction

Cavities can be found in many aerodynamic configurations such as landing-gear wells, junctions between surfaces and gaps in the metallic shell of an aircraft or a car. On a larger scale, the flow around channel gates or harbour entrances also shows cavity phenomena. These cavities have a strong influence on the local flow fields and the pressure distribution. The unsteady velocity, density and pressure fluctuation caused by the cavity can seriously damage the material. Large pressure fluctuations lead to buffeting, which is a response of the structure to these fluctuations or buffet, and so to structural failure. To reduce the drag and to avoid buffeting, control techniques are being developed. Therefore, the complex flow field over the cavity has to be investigated so one can understand what happens inside and around such geometry.

A lot of experimental and computational studies have already been performed to investigate the effect of the cavity. These studies were mainly based on compressible supersonic flows. Research to incompressible cavity flow has also been conducted, but especially for flow inside or around 2-dimensional cavities, or for flow inside 3-dimensional cavities known as lid-driven cavities. These cavities can be seen as a rectangular box with fixed bottom and side walls and a moving top wall. In such cavities, there is thus no interaction between the external flow and the re-circulating flow inside the cavity. In regular cavities, a shear layer forms between the external flow and the re-circulating internal flow. This unstable shear layer may flow over the cavity (open cavity), or may deflect inwards with a possible impingement on the floor (closed cavity). Cavities can thus be categorized into three types based on the cavity geometry [16]: open, closed and transitional.

- **Open cavities** have a length-to-depth ratio (L/D) less than 10. The shear layer spans the cavity and stagnates on the aft wall as can be seen in Figure 2.1.

- **Closed cavities** have a L/D greater than 13. The shear layer impinges on the cavity floor as can be seen in Figure 2.2. They are typically long and shallow.

- If the length-to-depth ratio L/D has a value between 10 and 13, the cavity is described as **transitional**. In this kind of cavity, either type of flow may occur, dependent on the inflow conditions.

```
Flow Direction

Flow Direction

Shear Layer

Recirculation Region

Aft Cavity Wall

Figure 2.1: Open cavity flow [8].
```
Other values or classifications are also used. Sarohia [39], for example, considers cavities with a depth-to-length ratio (D/L) less than one as shallow, while cavities with D/L greater than one are described as deep cavities.

Open cavity flow fields are very complicated due to the self-sustained shear layer oscillations that couple the internal and external regions. In order to understand this complex flow structure, experimental studies were performed on a 3-dimensional open cavity, assuming a rectangular geometry. This report describes these results of an incompressible flow past an open shallow cavity with a depth-to-length ratio (D/L) of 0.25 and a laminar Blasius boundary layer as inflow condition.

To start with, chapter three describes a literature survey of the flow in and around cavities. A description of boundary layers is given. What is a (laminar) boundary layer and how does it develop? After that, some boundary layer properties like the displacement thickness δ*, the momentum thickness θ and the shape factor H are introduced. The origin and behaviour of the turbulent boundary layer is described and based on the primary stability theory, the laminar to turbulent transition is further explained. The remaining of this chapter gives a summary of some numerical and experimental research projects on the flow field near or inside a cavity. The self-sustained oscillations of the flow past cavities are grouped into three types of interactions by Rockwell and Naudascher [34]. Also a brief general overview of previous studies on cavities is given. In order to understand the complex flow structure over and inside a cavity, some other research projects are described in more detail.

The method of hot-wire anemometry is described in chapter four. Hot-wire anemometry is an experimental method that is used to measure the mean and fluctuating velocity components in a flow. The basic principle of the system is that an incident flow will cool an electrically heated thin metallic element. The effective cooling velocity method as well as the calibration procedure are explained.

In chapter five the experimental setup at the School of Aeronautical Engineering at the Queen’s University of Belfast is described. Each of the following components will be discussed in detail: the wind-tunnel, the model, the transverse system, the hot-wire anemometer and the equipment for data acquisition and processing.

Chapter six describes the results of an unsteady incompressible laminar flow over the 3-dimensional open shallow rectangular cavity. Two visualization methods, miniature balls and a light reflecting fluid, are used to obtain a representation of the flow field inside the cavity. To see if there is a connection between these photographic results and the numerical values of the
velocity measurements acquired by using the method of single hot-wire anemometry, the boundary layer profiles outside the cavity are described. First the mean velocity boundary layer profiles are shown, then the following boundary layer properties are discussed: the boundary layer thickness $\delta$, the displacement thickness $\delta^*$, the momentum thickness $\theta$ and the shape factor $H$. To conclude the results outside the cavity, the pulsation intensity boundary profiles and values are presented.

Some numerical results obtained by Yao [48], [49] and [50] are presented in chapter seven. Some pictures are given based on the CFD method and on the LES approach. The experimental boundary layer profiles are also compared with the numerical CFD results.

To conclude, the conclusion of this experimental study is given in chapter eight.
3. Flow in and around cavities, a literature survey

3.1 Introduction

The theory described in this chapter is particularly taken from Schlichting [40]. This chapter starts with a description of boundary layers. What is a (laminar) boundary layer and how does it develop? After that, in section 3.3, some boundary layer properties like the displacement thickness $\delta^*$, the momentum thickness $\theta$ and the shape factor $H$ are introduced. The origin and behaviour of the turbulent boundary layer is described in 3.4. Based on the primary stability theory (section 3.6), the laminar to turbulent transition is further explained in section 3.5. The rest of this chapter gives a summary of some numerical and experimental research projects on the flow field near or inside a cavity. The self-sustained oscillations of the flow past cavities are grouped into three types of interactions by Rockwell and Naudascher [34]. These different interactions are described in section 3.7. A brief general overview of previous studies on cavities is given in 3.8. In order to understand the complex flow structure over and inside a cavity, some other research projects are described in more detail in section 3.9. Some of these results will be compared with the experimental work done in this study.

3.2 Boundary layers

The laws of mass, momentum and energy conservation as well as thermodynamic relations are used to describe the fluid motion. In case of an incompressible ($\rho = \text{cte}$) Newtonian fluid, only the following mass and momentum conservation equation are needed (Appendix A):

\begin{align}
\rho \mathbf{V} \cdot \nabla \mathbf{V} &= 0, \\
\rho \left[ \frac{\partial \mathbf{V}}{\partial t} + (\nabla \times \mathbf{V}) \mathbf{V} \right] &= -\nabla p + \mu \nabla^2 \mathbf{V},
\end{align}

(3.1)
(3.2)

with $\rho$ the fluid density, $t$ the time, $\mathbf{V}$ the velocity vector, $\bar{g}$ the gravitational acceleration, $p$ the pressure and $\mu$ the dynamic viscosity. Equation 3.2 is referred to as the Navier-Stokes equation.

Consider now a semi-infinite flat plate in a parallel flow with constant velocity $U_\infty$. Near the wall, the velocity profile will be disturbed as shown in Figure 3.1, due to the no-slip condition at the stagnant wall. This disturbed velocity profile, which grows in streamwise direction, is referred to as a boundary layer if its thickness $\delta$ is small compared to the distance of the leading edge $L$. In this boundary layer, the velocity decreases from the mean stream velocity $U_\infty$ at the edge of the boundary layer ($y = \delta$), to zero at the wall ($y = 0$).
The structure of such a laminar boundary layer along a flat plate has first been described by Blasius in 1908. First, the Reynolds number $\text{Re}_L$ is defined as the ratio of inertial to frictional forces:

$$\text{Re}_L = \frac{\rho U_\infty L}{\mu} = \frac{U_\infty L}{v}$$

(3.3)

with $v = \frac{\mu}{\rho}$ the kinematic viscosity. For $\text{Re}_L >> 1$, the steady boundary layer flow is described by the following set of equations (Prandtl equations):

- \[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0, \]
- \[ u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{dp}{dx} + \nu \frac{\partial^2 u}{\partial y^2}. \]

(3.4)

(3.5)

For a flat plate, it follows that $\frac{dU_\infty}{dx} = 0$ and, as a consequence, $\frac{dp}{dx} = 0$. The Prandtl equations 3.4 and 3.5 then reduce:

- \[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0, \]
- \[ u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \nu \frac{\partial^2 u}{\partial y^2}, \]

(3.6)

(3.7)

with the boundary conditions:

- $u = v = 0$ for $y = 0$
- $u \to U_\infty$ for $y \to \infty$. 

Figure 3.1: Boundary layer over a flat plate in a uniform flow [42].
Blasius found a solution for this problem. His method is explained in Appendix B. The reduced velocity profile \( U/U_\infty \) depends on a so-called similarity coordinate \( \xi \equiv y \sqrt{\frac{U_\infty}{v_x}} \). The solution \( U/U_\infty = \gamma(\xi) \) is shown in Figure 3.2.

![Figure 3.2: The velocity profile \( U(\xi)/U_\infty \) according to the Blasius solution [42].](image)

The boundary layer thickness delta \( (\delta_{0.99}) \) is defined as the distance to the wall where \( U = 0.99 U_\infty \). This corresponds with \( \xi = 4.9 \), so the Blasius boundary layer thickness \( y = \delta_{0.99} \) becomes:

\[
\delta_{0.99} = 4.9 \sqrt{\frac{v_x}{U_\infty}}. \tag{3.8}
\]

This expression can be written as a function of the Reynolds number (3.3) as follows:

\[
\frac{\delta_{0.99}}{x} = 4.9 \text{Re}_x^{-\frac{1}{2}}. \tag{3.9}
\]

An equation for the velocity profile of a laminar boundary layer over a semi-infinite flat plate has also been obtained by Pohlhausen. He assumed a fourth degree polynomial of the form

\[
\frac{U}{U_\infty} = A \left( \frac{y}{\delta_{0.99}} \right) + B \left( \frac{y}{\delta_{0.99}} \right)^2 + C \left( \frac{y}{\delta_{0.99}} \right)^3 + D \left( \frac{y}{\delta_{0.99}} \right)^4, \tag{3.10}
\]

as a valid representation of a laminar flow velocity profile. Using the boundary conditions and assuming a zero pressure gradient over the flat plate, the following equation has been derived (Appendix C):

\[
\frac{U}{U_\infty} = 2 \left( \frac{y}{\delta_{0.99}} \right) - 2 \left( \frac{y}{\delta_{0.99}} \right)^3 + \left( \frac{y}{\delta_{0.99}} \right)^4. \tag{3.11}
\]
The Pohlhausen approximation differs only a few percent from the Blasius solution. Therefore, equation 3.11 is used to plot the theoretical laminar boundary layer profile in chapter 6 to discuss the experimental results.

Now a formula for the laminar boundary layer profile and thickness has been derived, other boundary layer properties can be discussed.

### 3.3 Boundary layer properties

#### 3.3.1 Displacement thickness $\delta^*$

Since the definition of the boundary layer thickness is somewhat arbitrary, a physically more meaningful definition of the thickness is defined: the displacement thickness is the distance from the wall to an imaginary boundary which would have allowed the same quantity of fluid to pass if all the fluid outside this boundary had been flowing at the mean stream velocity $U_\infty$. The situation is illustrated in Figure 3.3. The displacement thickness $\delta^*$ is thus defined that:

$$\int_0^h u \, dy = (h - \delta^*) U_\infty,$$

with $h$ referring to a point far outside the boundary layer. The integral on the left side of 3.12 presents the real volume flux when the boundary layer is present, while the right side of the equation describes the flux for the non-viscous case with the wall displaced over a distance $\delta^*$.

For $h \to \infty$ the definition 3.12 becomes:

$$\delta^* = \int_0^\infty (1 - \frac{u}{U_\infty}) \, dy.$$  \hspace{1cm} (3.13)

This can also be seen as follows. Since the viscous force slows down the boundary layer flow, as a result, certain amount of the mass has been displaced (ejected) by the presence of the boundary layer (to satisfy the mass conservation requirement). The reduction in flow due to the boundary layer is also illustrated in Figure 3.3.

![Figure 3.3: The displacement thickness $\delta^*$ as the displaced amount of fluid [54].](image)

For a Blasius boundary layer over a flat plate, the value for $\delta^*$ becomes:

$$\delta^* = \int_0^\xi \left(1 - \frac{df}{d\xi}\right) \, d\xi \left(\frac{\nu}{U_\infty}\right)^{\frac{1}{2}} = 1.72 \left(\frac{\nu}{U_\infty}\right)^{\frac{1}{2}} = 1.72 \times Re_x^{-\frac{1}{2}}.$$  \hspace{1cm} (3.14)
3.3.2 Momentum thickness $\theta$

The momentum thickness $\theta$ is the distance from the wall to an imaginary boundary which would have allowed the same momentum to pass if all the fluid outside the boundary had been flowing at the free stream velocity $U_\infty$. It is defined in the same way as the displacement thickness, but now based on the momentum flux in the x-direction instead of the volume flux. In Figure 3.4, the momentum flux in A is equal to $\rho U_\infty^2 h$, in B it is given by:

$$
\int_0^{h+\delta^*} \rho u^2 dy = \int_0^h \rho u^2 dy + \rho \delta^* U_\infty^2 .
$$

(3.15)

![Figure 3.4: The displacement of streamlines outside the boundary layer over a distance $\delta^*$ [42].](image)

The difference between the fluxes in A and B, this is the loss of momentum caused by the boundary layer, is now defined as $\rho U_\infty^2 \theta$:

$$
\rho U_\infty^2 h - \int_0^h \rho u^2 dy - \rho \delta^* U_\infty^2 \equiv \rho U_\infty^2 \theta .
$$

(3.16)

Substitution of 3.12 for $\delta^*$ results in:

$$
\int_0^h (U_\infty^2 - u^2) dy - U_\infty^2 \int_0^h (1 - \frac{u}{U_\infty}) dy = U_\infty^2 \theta .
$$

(3.17)

So for $h \to \infty$, the momentum thickness $\theta$ becomes:

$$
\theta = \int_0^\infty \frac{u}{U_\infty} (1 - \frac{u}{U_\infty}) dy .
$$

(3.18)

For a Blasius boundary layer over a flat plate, the value for $\theta$ becomes:

$$
\theta = \int_0^\infty \left(1 - \frac{df}{d\xi} \right) \frac{df}{d\xi} d\xi \left( \frac{\nu}{U_\infty} \right)^{\frac{1}{2}} = 0.664 \left( \frac{\nu}{U_\infty} \right)^{\frac{1}{2}} = 0.664 \times Re_x^{\frac{1}{2}}.
$$

(3.19)
3.3.3 Shape factor $H$

A qualitative idea of the shape of a boundary layer velocity profile can be obtained from the dimensionless shape factor $H$. This is the ratio of displacement to momentum thickness:

$$H = \frac{\delta*}{\theta}.$$  \hspace{1cm} (3.20)

The shape factor is used to characterize the boundary layer:

- $H = 2.6$ for an undisturbed, constant-pressure, flat wall laminar boundary layer,
- $H = 1.3$ for a fully developed, constant-pressure, flat wall turbulent boundary layer.

3.4 Turbulent boundary layer

An exact stationary solution of equations 3.1 and 3.2 will only be obtained if the flow is stable. This means that small disturbances, which can occur at any time, have to attenuate in time. If these disturbances increase in time, however, a stationary solution cannot exist in nature. So the laminar boundary layer over a flat plate discussed in section 3.3 may become unstable and change to a turbulent boundary layer. An important parameter for the description of boundary layer stability is the Reynolds number $Re_5^*$ based on the displacement thickness $\delta*$:

$$Re_5^* = \frac{\rho U_\infty \delta^*}{\mu} = \frac{U_\infty \delta^*}{v}.$$  \hspace{1cm} (3.21)

The boundary layer over a flat plate will first always be laminar, until $Re_5^*$ has obtained a critical value. Then the boundary layer will change gradually in a transitional region from laminar to fully turbulent. This process is demonstrated in Figure 3.5.

![Figure 3.5: Transition from a laminar to a turbulent boundary layer over a flat plate [43].](image)

To investigate the stability of a laminar boundary layer profile, a harmonic spatial disturbance is supposed to be present with wave number $k$ ($k = 2\pi/\lambda$, with $\lambda$ the wavelength). What happens now with this disturbance? Does it attenuate, does it stay unchanged or does it increase in time? The solution is formed by solving the conservation laws for the linear disturbance [40]. The result of this linear stability analysis is presented in Figure 3.6. In the shaded area the boundary layer becomes unstable. The laminar boundary layer stays always stable for $Re_5^* < 520$. More downstream, for $Re_5^* > 520$, there is a range of wave numbers at which the disturbances will increase in time. This is responsible for the origin of turbulence.
In a turbulent boundary layer, the velocity $U$ fluctuates around an average value $\bar{U}(t)$. This average velocity is defined as:

$$\bar{U}(t) = \frac{1}{2t_0} \int_{t_0}^{t} u(t')dt',$$

with $2t_0$ a large time interval compared to the time scale of the turbulent fluctuations.

The velocity can thus be written as:

$$U(t) = \bar{U}(t) + U'(t),$$

with $\bar{U}(t)$ the mean velocity and $U'(t)$ the pulsation velocity.

To obtain a relation for the flow in a turbulent boundary layer, the following parameters are introduced:

$$\tau_0 = \mu \frac{\partial u}{\partial y} \big|_{y=0} \quad \text{the shear stress at the wall,}$$

$$u^* = \left( \frac{\tau_0}{\rho} \right)^{1/2} \quad \text{the friction velocity at the wall,}$$

$$y^* = \frac{y u^*}{v} \quad \text{the dimensionless height coefficient.}$$

Based on experimental results, the flow at the wall can be divided into three areas:

- the viscous sublayer with a linear velocity profile: $0 < y^* < 5$,
- the logarithmic wall layer with a logarithmic velocity profile: $20 < y^* < 1000$,
- the transitional area: $5 < y^* < 20$. 

Figure 3.6: Stability diagram of a laminar boundary layer over a flat plate [43].
For a semi-infinite flat plate, the logarithmic velocity profile is approximately described by:

\[
\frac{\bar{U}}{u^*} = 8.7 (y^+)^{1/7}.
\]  
(3.27)

From this result, the following formulas are derived (Appendix E). The velocity profile in a turbulent boundary layer is given by the Prandtl one-seventh power law:

\[
\frac{\bar{U}}{U_\infty} = \left( \frac{y}{\delta} \right)^{1/7}.
\]  
(3.28)

The growth of the boundary layer thickness \( \delta \) for a turbulent boundary layer along a flat plate is given by this equation:

\[
\frac{\delta}{x} = 0.37 \text{Re}_x^{-1/5}.
\]  
(3.29)

A turbulent boundary layer is thus thicker and grows faster than a laminar boundary layer (\( \delta/x = 4.9 \text{Re}_x^{-1/2} \)).

### 3.5 The laminar-turbulent transition

Before the boundary layer becomes completely turbulent, several processes take place during transition. This is demonstrated in Figure 3.7, based on experimental results. The boundary layer stays laminar till the Reynolds number reaches a certain value, \( \text{Re}_{x,\text{ind}} \), and the laminar to turbulent transition takes place. This indifference Reynolds number \( \text{Re}_{x,\text{ind}} \) is the smallest Reynolds number at which a neutral perturbation still exists. The start of transition takes place at the indifference point; the position of completed transition is called the critical point. For a flat plate with a sharp leading edge in a normal air stream, this critical point is situated at a distance \( x \) from the leading edge given by:

\[
\text{Re}_{x,\text{crit}} = \left( \frac{U_\infty}{\nu} \right)_{\text{crit}} = 3.5 \times 10^5 \text{ to } 10^6 
\]  
(3.30)

First, at \( \text{Re}_{x,\text{ind}} \), two-dimensional Tollmien-Schlichting waves are superimposed onto the laminar boundary flow. Primary stability theory, which is discussed in section 3.6, describes these TS waves. Further downstream, three-dimensional disturbances, caused by secondary instabilities, generate characteristic \( \Lambda \)-structures. These \( \Lambda \)-vortices form then turbulent spots, and downstream the boundary layer becomes fully turbulent as these spots fuse together. At \( \text{Re}_x = \text{Re}_{x,\text{crit}} \), the transition process is complete and the boundary layer has become turbulent.
3.6 Primary stability theory

In order to investigate the problem of boundary layer stability, the following procedure has been followed. A more detailed analysis can be found in Appendix F. The motion of a flow consists of a basic flow and a superimposed perturbation motion. By substituting this in the conservation laws and after linearization, linear equations for the perturbation components are obtained. Assume now a basic flow in the x-direction with velocity \( U(y) \) with a superimposed perturbation made up of single partial perturbations or modes. Each of these modes is a wave propagating in the x-direction. A stream function \( \psi \) can be introduced for the two-dimensional perturbation such that:

\[
\psi (x, y) = \varphi (y) e^{i(x - \lambda) t},
\]

with \( u' = \frac{\partial \psi}{\partial y} \) and \( v' = -\frac{\partial \psi}{\partial x} \).

Any disturbance can be described by these Fourier modes. The wavelength of the disturbance \( \lambda = 2\pi/\alpha \) is assumed to be real. The quantity \( \beta \) is complex:

\[
\beta = \beta_r + i \beta_i,
\]

with \( \beta_r \), the frequency of the mode, \( \beta_i \), the amplification factor.

If \( \beta_i < 0 \), the wave is damped and the flow stays laminar; for \( \beta_i > 0 \), the perturbation grows.
The ratio of the two parameters $\alpha$ and $\beta$ is also used:

$$c = \beta/\alpha = c_r + i c_i ,$$

(3.34)

where $c_r$ represents the phase velocity of the wave in the x-direction and $c_i$ is again the amplification factor.

The perturbation (3.32) has to satisfy the equations of motion. This results in the Orr-Sommerfeld equation:

$$(U - c)(\frac{d^2 \varphi}{dy^2} - \alpha^2 \varphi) - \frac{d^2 U}{dy^2} \varphi = -\frac{i}{\alpha \text{Re}_\delta} \left( \frac{d^4 \varphi}{dy^4} - 2\alpha^2 \frac{d^2 \varphi}{dy^2} + \alpha^4 \varphi \right).$$

(3.35)

All the parameters in this equation have been made dimensionless: the lengths have been divided by the boundary layer thickness $\delta$, the velocities are related to the maximum velocity of the basic flow $U_\infty$. Based on this equation, the boundary layer may become unstable at a certain Reynolds number. In Figure 3.8 curves of neutral stability ($\beta_i = 0, c_i = 0$) are shown both for $\beta_r$ and for $c_r$. The curves of neutral stability form the borders of regions in which the laminar boundary layer flow is unstable.

![Figure 3.8: Dependence of the curves of neutral stability for the perturbation frequency $\beta_r$ and the wave phase velocity $c_r$ on the Reynolds number for the boundary layer on a flat plate at zero incidence (Blasius profile). Theory according to W. Tollmien (1929); numerical computation by R. Jordinson (1970) [40].](image)

This type of instability, which is related to the viscosity, is called the Tollmien-Schlichting instability. As can be seen in Figure 3.8, the indifference point is situated at:

$$\text{Re}_{\delta, \text{ind}} = 520$$

(3.36)

This was already mentioned in section 3.4. There is also an upper limit for the characteristic magnitudes beyond which no further instabilities occur:

$$\frac{c_r}{U_\infty} = 0.39; \quad \alpha \delta^* = 0.36; \quad \frac{\beta_r \delta^*}{U_\infty} = 0.14$$

(3.37)
From these values, the smallest unstable wavelength can be calculated out of $\alpha \delta^*$:

$$\lambda_{\text{min}} = \frac{2\pi}{0.36} \delta^* = 17.5 \delta^* = 6 \delta_{0.99}$$  \hspace{1cm} (3.38)

Using equation 3.30 ($\text{Re}_{\text{crit}} = 3.5 \times 10^5$) and equation 3.14 ($\delta^* = 1.72 \left[ \frac{v}{U_w} \right]^{1/2}$), the critical Reynolds number, based on the displacement thickness $\delta^*$, at which the boundary layer has become completely turbulent, can be calculated:

$$\text{Re}_{\delta^*, \text{crit}} = 950$$  \hspace{1cm} (3.39)

This gives a larger value than the indifference point (3.36). The distance between the indifference point $\text{Re}_{\text{ind}}$ and the point $\text{Re}_{\text{crit}}$ at which the flow has become turbulent (determined experimentally), is dependent on the magnitude of the amplification of the unstable perturbations. This amplification is determined by the factor $c_i = \beta_i / \alpha$ as demonstrated in Figure 3.9.

![Figure 3.9: Curves of constant temporal amplification $c_i$ for the boundary layer at a flat plate at zero incidence in a large region of Reynolds numbers, after H.J. Obremski et al. (1969) [40].](image)

### 3.7 Flow over a cavity

A schematic presentation of the flow over a cavity is given in Figure 3.10. The incoming boundary layer separates at the front edge of the cavity, so a free shear layer develops containing spanwise instabilities. Depending on the dimensions of the cavity, the shear layer then reattaches either downstream the cavity or on its downstream end wall. This may result in oscillating disturbance waves which are acoustically and/or aerodynamically fed back to the original disturbance source at the leading edge of the cavity. This feedback loop builds up the disturbance waves at certain frequencies, with large oscillating pressure waves and noise as result.
Rockwell and Naudascher [34] classified three different types of oscillating flows according to the types of interaction involved in the oscillation:

- **Fluid Dynamic Interactions**: This type describes the coupling between the oscillations of the shear layer and the flow inside the cavity. The instability of the shear layer and the vortex shedding are the driving forces for the mechanisms of this flow. Especially the large scale coherent structures in the shear layer play a major role in these interactions. Many of the oscillating flows at low subsonic speeds over shallow cavities fall into this category.

- **Fluid Resonant Interactions**: The flow oscillations of this kind of interactions are controlled by the acoustic mode of the cavity. They occur for example in cavities with a large depth, in large volumes with small openings to the flow (Helmholtz resonators) or in flows over cavities at high Mach numbers.

- **Fluid Elastic Interactions**: This type involves the interactions between the shear layer and the elastic boundaries of the cavity. Loudspeakers which are used to drive cavity oscillations and the control surfaces on re-entry vehicles oscillating in phase with the separated flow field in their vicinity are some examples of this kind of flow.

These different types of interaction can occur together. This may happen for example in compressible flows over shallow cavities with deflectable surfaces or inside large cavities with high speed flows and baffles located within them.
3.8 History review of cavity flow studies

In this section a brief overview of experimental and numerical studies of cavity flows is given.

3.8.1 Experimental studies

In 1955, Karamcheti [14] and Roshko [36] reported their experimental results. Roshko investigated the time-averaged effects of 2-dimensional flows over a cavity at low Mach numbers with different cavity dimensions. He found that a single vortex existed inside a shallow cavity ($L/D > 1$) and small secondary vortices were present at the corners of the cavity. Karamcheti compared the characteristics of a laminar and turbulent boundary layer flow. For a turbulent boundary layer, the pressure fluctuation inside the cavity was much more violent. The peak amplitudes, however, were considerably less than in the case of a laminar boundary layer.

In the 1960’s, Plumbee et al. [31] investigated the acoustic response of large cavities in flows of Mach number from 0.2 to 5. This research was different from the earlier work, because he modelled the cavity as a 3-dimensional geometry. Later, Mauhl and East [20], and East [7] studied a 3-dimensional flow within deep ($L/D < 1$) cavities. Rossiter [37] did some subsonic and transonic tests ($0.4 \leq M \leq 1.2$) over shallow rectangular cavities. Triangular cavities at low Mach numbers were studied by Torda and Patel [47]. By investigating the flow-fields within the cavity, they found that a strong primary vortex stayed inside the cavity. Also small secondary vortices, as many as four, were seen near the bottom at the rear wall of the cavity. Deep cavities turned out to be much more stable than the shallow ones.

In the 1970’s, McGregor and White [21] studied the drag of rectangular cavities in supersonic and subsonic flow. The Mach number varied from 0.3 to 3. Heller, Holmes and Covert [11] investigated the pressure fluctuations inside the cavity. These spectra were obtained by conducting wind tunnel tests on cavities of $L/D$ ratio 4 to 7 over a Mach number range from 0.8 to 3. They also observed stronger spectral peaks for a laminar boundary layer.

Since the 1980’s, more and more research has been done into cavity flows with supersonic and transonic speeds. Stallings, Robert and Wilcox [46], for example, performed wind tunnel tests on supersonic flows past cavities. Plentovich et al. [30] investigated 3-dimensional cavity flow fields at subsonic and transonic speeds.

3.8.2 Computational studies

Computational Fluid Dynamics (CFD), a powerful tool for simulating complex flow fields, has been used to study the cavity flow field since the 1960’s. Earlier studies, like Weiss & Florsheim [51] and Pan & Acrivos [28], often assumed that the flows were steady. Nallasamy and Prasad [23] did more recent studies on steady cavity flows at high Reynolds number (between 0 and 50,000). They found that three fully viscous eddies were formed inside the cavity. Also the variation of the steady cavity flow field with Reynolds number was investigated.

Typical studies of the computation of unsteady incompressible cavity flows have been done by Mehta and Lavan [22], O’Brien [26], Wood [52] and by Pereira and Sousa [29]. The flow in a 2-dimensional channel with a rectangular cavity is calculated by Mehta and Lavan. O’Brien studied closed streamlines associated with channel flow over a cavity. He calculated the viscous Stokes flow in a rectangular cavity with parallel shear flow by a direct finite difference technique. A multi-grid approach was used by Wood to investigate 2-dimensional viscous flows within rectangular cavities. Pereira and Sousa calculated the computation of 2-dimensional unsteady flow past a rectangular cavity. All these studies deal with 2-dimensional cavity flows.
So far, little investigation has been carried out into 3-dimensional cavity flows. It is limited to the computation of the flows inside a lid-driven cavity [6], [13], [17], [18], [19].

More computational research has been done into compressible cavity flows. The flow within and past a cavity, for both 2- and 3-dimensional cavities, has been investigated. Borland [4] computed a solution for the 2-dimensional Euler equations for the time-dependent inviscid compressible flow over a 2-D cavity. Pressure oscillations in an open cavity were analyzed by Hankey and Shang [10] using the 2-D NS equations. The Mach number inside the cavity reached a value of 0.5 with a supersonic flow outside the cavity. Due to the evolution in computation and storage, complex 3-D models can be simulated since the 1980’s. Gorski, Ota and Chakravarthy [9] for example, calculated the 3-D flow fields with laminar and turbulent flows past an irregular cavity. Also the 3-D transonic flows past open and transitional cavities were studied by Srinivasan, Baysal and Plentovich [45]. Rizzetta [32], to conclude, calculated 3-dimensional supersonic flows over a rectangular cavity.

### 3.8.3 Topic of present study

As a summary, the previous studies on cavity flow fields can be grouped into four major areas:

1. Two-dimensional incompressible flows inside or past a cavity,
2. Two-dimensional compressible flows inside or past a cavity,
3. Three-dimensional incompressible flows inside a cavity,
4. Three-dimensional compressible flows inside or past a cavity.

So far, little research has been carried out into 3-D incompressible flows past cavities. Nevertheless, a three-dimensional incompressible flow past a cavity represents an important class of flow problems. Therefore, H. Yao [48], [49], [50] performed a CFD analysis of such flows. This report describes experimental verification of Yao’s results.

### 3.9 Related research projects

#### 3.9.1 Research on 2-D cavities

Rossiter [37] did some subsonic and transonic tests \((0.4 \leq M \leq 1.2)\) over shallow rectangular cavities. He discovered that the unsteady pressures acting in and around the cavity contain both random and periodic components. The random component predominates in shallow cavities \((L/D > 4)\), whereas the periodic component predominates in deep cavities. He concluded that the acoustic resonance within the cavity is responsible for the periodic pressure fluctuations. These fluctuations can be suppressed by fixing a small spoiler ahead of the cavity as shown in Figure 3.11.

![Figure 3.11: Spoiler positioned at leading edge of cavity [37].](image-url)
Pereira and Sousa [29] did experimental (using a water tunnel) and computational research to the unsteady flow past a rectangular cavity for a Reynolds number, based on the cavity depth, of $Re_D = 3,360$. The cavity has a length-to-depth ratio of $L/D = 2$. They could see that the destabilization of the flow field is caused by the interaction of the unstable shear layer with the recirculating flow field. The shear layer is thus forced to adapt to the flow structure imposed by the Kelvin-Helmholtz instability. Pereira and Sousa explain the physical mechanism describing the observed flow oscillations as follows. The instability process can be interpreted as a shear layer destabilization of the recirculating flow field. It is reasonable to conclude that the stable part of the flow (the recirculation eddies), due to their larger characteristic energy, determine the nature and thus the frequency of the oscillations. The unstable part of the flow (the shear layer), however, locks onto it. The oscillation of the shear layer, due to the recirculation region or to the upstream influence, is responsible for the variability of events in unsteady cavity flows.

A visualization of the flow inside an open shallow cavity with a length-to-depth ratio $L/D = 3.5$ at a Reynolds number of $Re = 11,300$ was made by Neary and Stephanoff [24]. The result is presented in Figure 3.12. At very low Reynolds numbers, a single vortex is centred slightly downstream of the middle of the cavity and fills almost the entire cavity. At higher Reynolds numbers, this vortex shifts downstream the cavity and if the Reynolds number keeps increasing, fluid along the bottom wall separates and a small counter clockwise rotating second vortex forms. This separation occurs because the fluid along the wall decelerates at an increasing rate. Figure 3.12 also shows a tertiary vortex, which is generated by the strong shear in the flow upstream of the primary vortex. Since the tertiary and the primary vortices rotate in a clockwise direction, a saddle point exists in the flow field between their cores.

![Figure 3.12: The fluid motion in the cavity at Re = 11,300. The shear layer is stable and there are three vortices in the cavity. (a) Photograph of the flow. (b) Sketch of the flow. [24]](image)

From experimental results, Neary and Stephanoff [24] classified the type of fluid motion depending on the Reynolds number into three regimes. In the first regime, regime I ($Re = 31,300$), the time trace from a pressure transducer, placed at the downstream side of the cavity,
varies weakly in amplitude. Its frequency spectrum shows a first peak, its first harmonic, and a second frequency. The pressure time trace is intermittent in regime II (Re = 31,900) and the two frequencies are further apart than in regime I. The first frequency is due to the shear layer instability, the second is believed to depend on a transverse wave in the primary cavity vortex. If these waves are constructively interfering, the exchange of fluid between the vortex and the free stream is enhanced. They even observed fluid from the primary vortex to burst through the shear layer, with a period of apparently random motion as a result, if the two waves are in phase and the amplitude of the transverse wave is large enough. At a higher Reynolds number, at regime III (Re = 33,500), the pressure oscillations vary strongly with time and include frequent periods of intense irregular behaviour. The pressure cycles may have double peaks when the vortices formed in the shear layer partially clip at the downstream edge of the cavity. This clipping does not, however, coincide with a decay in the shear layer oscillations, as it does in regime II. The frequency spectra for the different regimes are given in Figure 3.13.

Figure 3.13: Pressure signal and frequency spectra for a period of pressure oscillations of (a) Regime I, (b) Regime II (shear layer oscillations are increasing with time), (c) Regime II (shear layer oscillations are decreasing and recovering with time) and (d) Regime III (with double peaks of the pressure signal) [24].
Mehta and Lavan [22] performed a computational research on the flow in a 2-dimensional channel with a rectangular cavity. They solved the NS equations for a laminar incompressible flow in terms of the stream function and vorticity for Reynolds numbers from 1 to 1,500 and cavity length-to-depth ratios L/D of 0.5, 1 and 2. The solutions for the vortex flow in a square cavity seemed to be time-dependent for a Reynolds number of 10. As the Reynolds number increased, they saw that the strength of the vortex increased and then decreased. The vortex moved downstream and upward, causing the shear layer to become thin.

The unsteady two-dimensional flow upon the downstream corner is studied by Rockwell and Knisely [35]. The Reynolds number, based on the cavity length, varies from Re = 1.9*10^4 to Re = 4.6*10^4. They observed the eddy shedding caused by the destabilization of the recirculating flow field, as described in the previous section. The shear layer locks onto the formed disturbance imposed by the stable part of the recirculating flow field. However, if this shear layer cannot adapt to the disturbance, the shear layer will itself rule over the remainder of the flow. The complex process of interaction between the vertical structures and the impingement corner is referred to as jitter. Several interactions of the shed vortex with the impingement corner can occur as demonstrated in Figure 3.14. The vortex may suffer clipping by the impingement edge in a complete or in a partial way, or else escape from being clipped.

Figure 3.14: Vortex impingement stages; (a) vortex approaching cavity corner, (b) complete clipping, (c) partial clipping and (d) escape of vortex [35].
3.9.2 Research on 3-D cavities

Maull and East [20] investigated three-dimensional effects. They concluded that a large ratio of width-to-length W/L does not necessarily result in effectively two-dimensional flow. Rizzetta [32] computationally studied the unsteady flow over a 3-dimensional cavity at a free stream Mach number of 1.5 and a Reynolds number of Re = 1.09*10^6. The following conclusions followed:

- The 2D predictions for the mean static pressure were in very good agreement with the centreline prediction of the 3D computations.
- The predicted resonant frequencies from both 2D and 3D computations were practically equal.
- The level of unsteady pressure measurements was slightly overpredicted by the 2D calculations compared to the 3D.

This confirmed that the open cavity flow field, comprising of a disturbance amplification in the shear layer accompanied by a feedback to maintain the flow field oscillation, is two-dimensional in nature.

3.9.3 Varying the width of the cavity

The effect of changing the length-to-width ratio L/W is studied by Block [3]. The Mach number ranged from 0.1 to 0.5. The length-to-depth ratio L/D varied from 0.3 to 8.0 and the length-to-width changed from 0.3 to 1.85. From her results she deduced that the sound power levels and the ratio of the centre frequency to the frequency bandwidth of the peak were both increased by decreasing the cavity width. The resonance frequencies didn’t change despite the varying dimension.

Ahuja and Mendoza [1] investigated the behaviour of the shear layer for a three-dimensional cavity. They found that such a cavity cannot maintain a coherent shear layer across its width because of the end effects that cause the flow to spill over the sides of the cavity. By decreasing the width of the cavity, they noticed that the flow behaviour became chaotic in the outer regions near the leading edge of the cavity. As a result, these end effects change the spanwise coherence of the excited instability waves in the mixing layer, which is likely to change the amplitude of the cavity tones. If the length-to-width ratio L/W < 1, the flow appears to be two-dimensional whereas the flow tends to be three-dimensional for L/W >1.
4. Hot-wire anemometry

4.1 Introduction

This description of the hot-wire anemometry technique is particularly taken from Bruun [5]. Hot-wire anemometry is an experimental method that is used to measure the mean and fluctuating velocity components in a flow. The basic principle of the system is that an incident flow will cool an electrically heated thin metallic element. The temperature change is a function of the velocity magnitude and the flow angle to the sensor. There are two methods of controlling the sensor's heating current: the 'constant current anemometry' (CCA) and the most commonly used 'constant temperature anemometry' (CTA). Using the CTA set-up, the hot-wire is placed in a Wheatstone-type bridge circuit. A feedback loop controlling the voltage supply $E_B$ of the bridge maintains the wire at a constant temperature. The current to the wire is changed by a variation of the voltage supply in such a way that it compensates for the cooling of the wire. This voltage $E_B$ can be used as a measure of the time dependent flow velocities.

There are many different probe configurations used in hot-wire anemometry, depending on the nature of the flow to be studied. A few of these probe configurations are presented in Figure 4.1.

![Figure 4.1: Hot-wire probe configurations [5].](image)

Single wires are normally used for velocity measurements of one mean direction. However, slanted wires can be used to measure all of the velocity components by rotating the probe around its axis. X-wire probes are used mainly in 2-dimensional flows to measure two components of the instantaneous velocity vector at a plane between the two wires and also to measure the vorticity corresponding to these two components. They can therefore be used to map 3-dimensional flows by measuring in two different planes, with each plane yielding two components of the velocity vector. Triple-wire probes measure three instantaneous velocity components simultaneously. However calibration is much more complicated than for a single wire probe. Therefore a single normal hot-wire anemometer will be used for this experiment.
4.2 Effective cooling velocity method

The heat exchange between a thin heated cylinder and the flow around the cylinder is dependent on the velocity vector with respect to the cylinder. This is called the effective cooling velocity.

The velocity components are defined according to Figure 4.2: $U_N$ (normal to the wire in the direction of the main flow), $U_T$ (tangential) and $U_B$ (bi-normal, normal to $U_N$ and $U_T$). These components are presented in Figure 4.2.

![Figure 4.2: Velocity components](image)

The effective cooling velocity equals:

\[ V_e = V \cos \alpha = \sqrt{U_N^2 + U_B^2}, \]  
(4.1)

where $\alpha$ is the yaw angle of the flow with respect to the wire.

In reality hot-wires have a finite length and lose heat through the support prongs, so the relationship given by Hinze [12], which also takes into account the tangential component of the effective cooling velocity, is more often used:

\[ V_e^2 = U_N^2 + U_B^2 + k^2U_T^2 \]  
(4.2)

or,

\[ V_e^2 = V^2(\cos^2 \alpha + k^2 \sin^2 \alpha) \]  
(4.3)

where $k$ is the yaw coefficient. This term can be determined through calibration and varies between 0.1 and 0.3. In this study, the contribution of $U_T$ to the signal is neglected ($k = 0$).
4.3 Calibration procedure

The calibration procedure consists of two parts. The first part deals with setting the parameters in order to obtain the best frequency response. During this procedure, the filters and gain are set, and the resistance of the wire is measured. The Square Wave Test, described in section 4.3.1, is used to achieve this. The second part of the calibration procedure is the calibration of the single hot-wire probe and is described in section 4.3.2.

4.3.1 The square wave test

The square wave test allows measurement and optimization of the system frequency response. For this test, a square wave generator is placed parallel to the probe, and then a small-amplitude square wave is added to the sensor heating current. This is illustrated in Figure 4.3. The sensor can be operated in steady flow to allow the square wave signal to be superimposed on a large mean signal.

![Wheatstone bridge with square wave generator](image)

Figure 4.3: Wheatstone bridge with square wave generator [15].

The square wave test is based on the assumption that heating and cooling of the sensor by varying the fluid velocity is thermodynamically identical to heating and cooling of the sensor by varying the heating current. As convective cooling is a surface phenomenon, while electrical heating takes place in the sensor material, these two phenomena are only approximations of one another.

When the square wave current is introduced, the sensor current will go high and then drop to its original value as the feedback amplifier reduces the heating current to balance the bridge. A sudden decrease in fluid velocity has the same effect, as shown in Figure 4.4. Alternatively, when the square wave current goes low, the feedback amplifier will increase the heating current to bring the sensor resistance back to its original value.
The time constant for the system can be calculated by measuring the time required for the output voltage to drop from its peak amplitude to an amplitude that is 63.2% less, as shown in Figure 4.6. In this output signal, several features such as rise time, overshoot and oscillations can occur. The overshoot is a kind of response peak which can completely disappear when the system is overdamped (Figure 4.5 a). Underdamping gives rise to oscillations (Figure 4.5 c) which can be very damaging in a system with feedback. An overdamped system, however, has a slow response or a long rise time, which narrows the system bandwidth and hence the speed with which it can respond to changes in the input is limited. Therefore the damping has to be adjusted to an optimum point by changing the gain and filter settings to allow a response time as short as possible without producing oscillations.

Figure 4.5: Typical anemometer pulse responses [27].
4.3.2 SN-Probe calibration

This procedure is used to determine the sensitivity coefficients of the wires (associated with their electrical set-ups) with respect to the different flow variables. The variables are those quantities to which the wires would directly respond when submitted to an unsteady situation. Thus, for a given fluid and for a wire normal to the flow, the variables are the velocity and the ambient temperature at low speed. The flow variables are measured by conventional equipment (Pitot tube, thermocouple, etc.). The anemometer output voltages are related to the flow velocity through the 'effective cooling velocity'. The calibration curves (output voltage versus velocity) are obtained and described by a polynomial relationship. The probe is calibrated using the "steady-state" method, in which the probe is held stationary while the fluid passes over it by using the Taylor wind tunnel. The anemometer output voltage is recorded at discrete velocities starting from 20 m/s down to zero.

A calibration curve of output voltage versus velocity is used to display the calibration data. Figure 4.7 shows an example of such a calibration curve. TableCurve software is used to find the polynomial relationship between voltage and velocity that best fits the calibration points. Polynomial relationships of power 9 or 10 are the best fitted for the range of the current experiments. The polynomial relationship is then applied to the acquisition program to process data during acquisition.
A small remark on the power of the calibration curve has to be made. Since this a polynomial of power ten, fluctuations are expected to arise outside the described range, or even between the given calibration points. After closer investigation, no fluctuations or deviations from the calibration curve were found, so this kind of calibration curve could be used. This is presented in Figure 4.8. The x-increment for the voltage is set at 0.0001.
Figure 4.8: The calibration curve plotted for a) the range of the experiment (4.5 V), b) a larger range (10 V) and c) a very small range (0.004 V).
Due to ambient temperature changes, the bridge voltage $E_B$ is corrected as follows: a reasonable assumption is that $\frac{E_B^2}{(T_S - T)}$ is constant for a given velocity as temperature changes. Therefore, a bridge voltage $E_B'$ can be predicted for a new temperature $T'$, as follows:

$$
E_B' = E_B \left( \frac{T_S - T'}{T_S - T} \right)^{\frac{1}{2}},
$$

(4.4)

with: $E_B'$: the corrected bridge voltage,
$E_B$: the measured bridge voltage,
$T_S$: the temperature of the probe,
$T'$: the reference temperature,
$T$: the measured temperature.

The temperature correction is applied and each set of data is collected and processed after the correction. The value for the probe temperature $T_S$ is obtained using the following formula:

$$
T_S - T = \frac{R_S - R}{\alpha R},
$$

(4.5)

with: $\alpha = 0.0036$ [°C⁻¹] the resistivity coefficient,
$R_S$ the resistance of the heated wire at $T_S$,
$R$ the resistance of the wire at $T$.

A recommended value for the overheat ratio $R_S/R$ for a tungsten wire is 1.8. So formula 4.5 results in:

$$
\Delta T = T_S - T = \frac{\Delta R}{\alpha R} = 222 \, ^{\circ}C.
$$

(4.6)

For these experiments, the temperature of the probe is set at $T_S = 222 \, ^{\circ}C$. 

34
5. Experimental setup

5.1 Introduction

In this chapter the experimental setup at the School of Aeronautical Engineering at The Queen’s University of Belfast is described. A schematic overview of the experimental test facility is given in Figure 5.1.

![Image of schematic overview of experimental setup]

Figure 5.1: A schematic overview of the experimental setup.

Each of the following components will now be discussed in detail: the wind-tunnel, the model, the transverse system, the hot wire anemometer and the equipment for data acquisition and processing.

5.2 The wind-tunnel

All experimental work has been carried out in the School of Aeronautical Engineering low speed wind-tunnel which is presented in Figure 5.2. This wooden wind-tunnel of the return circuit type has a rectangular test section (Figure 5.3) which is 1.8 m long, 0.84 m high and 1.14 m wide. The maximum velocity at the test section is 45 m/sec (the majority of tests were carried out in the range of 10 – 20 m/sec). The operating temperature of the tunnel during the experiments was approximately 23°C.

The temperature at the test section is monitored by a Fluke 52 thermometer with K-type thermocouple. To accurately adjust the speed settings of the wind-tunnel, a Betz manometer was used. This instrument measures the difference between the wind-tunnels pitot and static pressure probes and displays the result in millimetres of water.
Figure 5.2: The low speed wind-tunnel at the School of Aeronautical Engineering.

Figure 5.3: A picture of the test section with the transverse system.
5.3 Flat plate model with the cavity

A picture of the Perspex model is given in Figure 5.4.

![Figure 5.4: A photograph of the front (a) and rear (b) view of the Perspex model.](image)

The flat plate of the model has a length of 1,320 mm, a width of 700 mm and a thickness of 24.3 mm. The cavity is situated at 300 mm from the leading edge. It measures 97.2 mm in length and the width measures 291.6 mm. The cavity is 24 mm deep so the depth-to-length ratio $D/L = 0.25$ and the width-to-length ratio $W/L = 3$. This means this is an open shallow cavity. On one side of the cavity some pressure taps were provided. These are not only positioned inside the cavity, as can be seen in Figure 5.7, but also on the leading edge (8) for controlling the stagnation point and lengthwise (19) along a diagonal over the whole model. A schematic overview of the model and its dimensions (in mm) is presented in the Figures 5.5 to 5.7.

![Figure 5.5: Top view of the model with the cavity and the flap at the right end (length in mm).](image)
To ensure that the flow separation is prevented at the junction between the nose section and the pre-cavity flat plate section, the nose section was shaped as an ellipse as shown in Figure 5.6. For the ellipse in the upper part of the model, the length of the major axis is 100 mm; for the ellipse in the lower part, the major axis is 386 mm. The geometry was chosen as such because a smooth pressure distribution was desired near the leading edge. The shape of the leading edge minimized the pressure gradients on the working side in the region near the stagnation line, and prevented flow separation near the leading edge.

Initial experimental tests in another research project showed a variation in the static pressure across the model, in the neighbourhood of the cavity. So a trailing edge flap was attached to the model. The flap can be adjusted to fix the stagnation point and to ensure a zero pressure gradient across the cavity. Another benefit of the inclusion of the flap is that it increases the downstream length of constant pressure and thus minimizes the trailing edge effects intruding on the flow characteristics of the cavity. The flap is 201 mm long, has a width of 700 mm and a varying thickness from 24.3 mm at the side of the trailing edge of the model to 3 mm (rounded edge) at the trailing edge of the flap.

The profile of the dynamic pressure $p_0 = \frac{1}{2} \rho U^2$ above the model in the stream wise direction is shown in Figure 5.8 for different positions of the flap. If the flap is turned completely down (-90° with the model) there certainly does not exist a zero pressure gradient over the model. In the other cases, where the flap makes an angle of -5°, 11° and 15° with the model, a zero pressure gradient over the cavity (situated between 0.3 and 0.4 m) is obtained. These were the only positions at which the trailing edge flap could be fixed. Based on these measurements, it was decided to fix the flap angle at 11°.
Figure 5.8: The dynamic pressure profile over the model for different positions of the trailing edge flap.

5.4 Transverse system

A transverse mechanism is used to take accurate readings with the hot-wire system. The probe has to move vertically in small steps to obtain reliable results. The transverse system is mounted on top of the test section of the Taylor Wind-Tunnel. A symmetric aerofoil, that enters the test section of the wind tunnel to reduce the force on the probe holder, is fixed on the vertical column. The vertical position of the probe can be controlled automatically; the smallest step size being 10 \( \mu \text{m} \).

The system is also movable in the x-direction (streamwise direction); this can be carried out manually with set stations in the x-axis. Therefore the vertical column was mounted in a set of rails and a 30 mm wide slot runs stream wise the full length of the false roof of the tunnel.

To move the probe in the z-direction, across the test section, an extension is adjusted in the bottom of the aerofoil. It consists of a thin steel plate, 20 mm wide, 100 mm long, and 2 mm thick, attached perpendicularly to the chord of the aerofoil. It can be moved manually. A picture of the upper part of the transverse system is given in Figure 5.9 a. The lower part of the transverse system with the extension is shown in Figure 5.9 b.
5.5 Hot-wire anemometer

For our measurements a miniature boundary layer single wire probe (DANTEC 55P15) has been used. The probe utilizes a tungsten wire with 5 µm diameter and 1.25 mm length. A picture of such a probe is shown in Figure 5.10.

The miniature boundary layer probe is connected to a DISA hot-wire anemometer system operating in CTA mode, with a bridge ratio of 1:20 and an overheat ratio of 1.8. This system consists of a Main Unit (55M01), a Constant Temperature Anemometer (CTA) Standard Bridge (55M10), a Voltmeter (55D31), a RMS Unit (55D35), and a Signal Conditioner (55D26).
The hot-wire anemometry technique is described in detail in the previous chapter, here follows just a brief explanation of these components:

- The Wheatstone bridge circuit sensitively adjusts the voltage through the hot-wire.
- The Voltmeter is used to check the normal operation of the probe.
- The RMS Unit provides a quick method of assessing the turbulence at the hot-wire.
- The Signal Conditioner is in fact a filter and is used to reduce the interference from electrical noise pollution. In this case the high pass filter was set at 5 Hz and the low pass to 5 kHz.

Velocity measurements have to be taken as close to the surface as possible and, what is most important, the distance between the start position of the probe and the surface has to be the same for all measurements. Therefore the following technique, as presented graphically in Figure 5.11, is used. A 9V battery is connected to a thin aluminium foil, which can be seen in Figure 5.10. The battery is also connected to the prongs of the probe. Now the probe is slowly lowered towards the surface. The computer checks the voltage between the probe and the aluminium foil each time before the probe is lowered. Once the probe touches the surface, the voltage drops and the program stops the transverse system. Contact is achieved due to the closed circuit of the battery, the aluminium foil and the prongs. Now the probe is raised 0.1 mm from the surface before measurements are started. At this way, each measurement starts at the same reference height.

![Battery circuit used to set the height reference.](image)

**Figure 5.11: Battery circuit used to set the height reference.**

### 5.6 Data acquisition

The probe signal is split into two parts: the mean (DC) and the unsteady (AC) component. The computer will average out the readings corresponding to the DC probe voltage. The signal used for this purpose is supplied directly to the acquisition board input without any filtering and the range of this channel is set at ± 10 V. The digital voltmeter attached to the mean line is only used as a check. The fluctuations however, will be filtered, but before that takes place the line is split and a RMS unit is installed to give an indication of turbulence within the flow. The high-pass frequency filter was set at 5 Hz to remove the DC signal and the low frequency disturbances of the signal (tunnel noise, motor pumping, vibration etc.) and the low-pass frequency filter was
set at 5 kHz to prevent aliasing of the signal. The range for the AC component, the fluctuating voltage, is set at ± 0.5 V.

These signals are then acquired by a PC using a National Instruments acquisition card (NI PCI-6035E), while a LabView code takes and carries out analysis of the readings. With the Easy Tools software the transverse system could be controlled through LabView. With a program it is possible to move the system, acquire data from the hot-wire anemometer, and repeat the procedure for a predefined number of steps.
6. Experimental results

6.1 Introduction

This chapter describes the results of an unsteady incompressible laminar flow over a 3-dimensional open shallow rectangular cavity. The cavity has a depth-to-length ratio $D/L = 0.25$, and a width-to-length ratio $W/L = 3$. The velocity of the incoming flow is set to 15 m/s to meet the laminar Blasius boundary layer inflow condition. Also a zero pressure gradient condition across the cavity is obtained by the trailing edge flap at the end of the model as discussed in section 3.3.

In section 6.2 two visualization methods, miniature balls and a light reflecting liquid, are used to obtain a representation of the flow field inside the cavity. To see if there is a connection between these photographic results and the numerical values of the velocity measurements acquired by using the method of single hot-wire anemometry, section 6.3 describes the boundary layer profile outside the cavity. First the mean velocity boundary layer profiles are shown, then the following boundary layer properties are discussed: the boundary layer thickness $\delta$, the displacement thickness $\delta^*$, the momentum thickness $\theta$ and the shape factor $H$. To conclude the results outside the cavity, the pulsation intensity boundary profiles and values are presented.

6.2 Visualization methods

To obtain a first impression of the flow field inside the cavity, some visualization methods are used. Two methods are described in this part: miniature balls and a light reflecting liquid.

6.2.1 Miniature balls

These balls, with a diameter of 2 mm, are placed randomly inside the cavity. The velocity is set at 20 m/s. Immediately after the wind tunnel is switched on, the balls move to the upstream side of the cavity. Due to the height of the balls and the high velocity, the balls will move to a position very close to the upstream edge. A separation line is formed between the upstream side, the area where the balls are, and the downstream side. This border line is not supposed to be straight, because several structures and vortices are present inside the cavity, especially at the upstream side. If the balls are set in motion with a stick, they move to the upstream side and come back to the border line. At the corners a circular motion was found: the balls move to the side, then to the upstream side where after they come back to the border line. A picture taken at the end of the experiment is presented in Figure 6.1.
6.2.2 Light reflecting liquid

In the next visualization technique a light reflecting liquid is spread out over the bottom of the cavity. The viscosity of the liquid is such that the interaction between the bottom and the vortices can be visualized. During the experiment photographs were taken with the help of UV-light. The result is shown in Figure 6.2 a, b and c.
Immediately after the wind tunnel is switched on at 15 m/s, one can see that the liquid moves from the downstream to the upstream side of the cavity. It creates a straight border, in contrast to the balls, that doesn’t move anymore as time proceeds. At the upstream side however, the liquid is pushed by the vortices and structures of the cavity, and a symmetric pattern can be observed. After 180 seconds, this pattern doesn’t change anymore; this means that at the bottom of the cavity a stable and symmetric situation occurs. A vortex can be seen at each corner, which is responsible for the behaviour of the balls described before. The pattern is not completely symmetric, but this can be caused by the small loss of liquid through the pressure taps at the left side of the cavity.

The streamwise flow pattern over the shallow open cavity and the vortex structure at the upstream corners inside the cavity are given in Figure 6.3.
6.3 Results outside the cavity

In order to understand the flow around and inside the cavity, velocity measurements at different positions have to be taken. Therefore the method of single hot-wire anemometry is employed as described in chapter 4. A Labview program calculates the mean velocity and the pulsation velocity from the signal coming from the probe (in V). The signal used to calculate the pulsation velocity is filtered by a low-pass filter (5 Hz) and a high-pass filter (5 kHz). To calculate the mean and pulsation velocity from the mean and pulsation voltage, the equation derived from the calibration curve is applied.

The velocity measurement is started as close to the surface as possible and goes up in steps till outside the boundary layer. The number and size of the steps is dependent on which strip the measurement is taken. Velocity profiles are obtained at different lateral positions (9 in total) for each strip. These positions are the cross points off the metallic strips ... and the ---- lines as shown in Figure 6.4. 5,000 samples (sample rate: 1,000 per second) are taken for the mean velocity profile and 100,000 samples (sample rate: 10,000 per second) for the pulsation profile. The number of samples for the pulsation profile is much higher than for the mean velocity profile because of the fluctuations in the pulsation profile. The sample rate is set at 10,000 per second for a maximum frequency of 5 kHz.

The horizontal aluminium foil strips are attached at the following positions (distance between the centre of the strips and the leading edge):

- Strip 1: $x = 255$ mm
- Strip 2: $x = 295$ mm
- Strip 3: $x = 402$ mm
- Strip 4: $x = 472$ mm
- Strip 5: $x = 612$ mm

Strip 1 and 2 are in front of the cavity, strip 2 is attached just before the cavity. Strip 3 is fixed just after it, and strip 4 and 5 are further away from the cavity to see how the velocity profile evolves.
The positions at which the measurements are taken are (middle of the cavity is 0):

- Position 1: $z = -235$ mm
- Position 2: $z = -150$ mm
- Position 3: $z = -140$ mm
- Position 4: $z = -72$ mm
- Position 5: $z = 0$ mm
- Position 6: $z = 72$ mm
- Position 7: $z = 140$ mm
- Position 8: $z = 150$ mm
- Position 9: $z = 235$ mm

Through the symmetric pattern of these positions, a comparison between the corresponding points can be made. Position 1, 2, 8 and 9 are outside the cavity (2 and 8 just next to it), the places 3 and 7 are situated just inside the cavity. Position 5 corresponds with the middle of the cavity.

6.3.1 Boundary layer profile for mean velocity

Boundary layer measurements were taken at each position for the 5 strips. To compare the mean velocity profile at each strip, all the spanwise positions for a strip are plotted in the same picture. These results are shown in Figures 6.5 to 6.9 for each strip. The profiles are plotted as the dimensionless velocity $u = U/U_{99}$ (the velocity $U$ divided by 99% of the maximum velocity.
$U_{\infty}$) versus the dimensionless height $\eta = y/\delta_{0.99}$ (the height $y$ divided by the thickness of the boundary layer $\delta_{0.99}$). The theoretical profiles for a laminar and a turbulent boundary layer are also presented in these plots. The laminar profile uses the Pohlhausen equation:

$$u_{\text{lam}} = 2\eta_{\text{lam}} - 2\eta_{\text{lam}}^3 + \eta_{\text{lam}}^4$$

For the turbulent boundary layer, the Prandtl one-seventh power law is used:

$$u_{\text{turb}} = \eta_{\text{turb}}^{1/7}$$

Figure 6.5: Mean velocity boundary layer profile for the different positions at strip 1.
Figure 6.6: Mean velocity boundary layer profile for the different positions at strip 2.

Figure 6.7: Mean velocity boundary layer profile for the different positions at strip 3.
Figure 6.8: Mean velocity boundary layer profile for the different positions at strip 4.

Figure 6.9: Mean velocity boundary layer profile for the different positions at strip 5.
Upstream the cavity, at strips 1 and 2, the boundary layer is laminar for each position. The profiles are similar to the theoretical laminar boundary layer as shown in Figure 6.5 and 6.6. At strip 3, Figure 6.7, a transitional profile occurs. The boundary layer is neither laminar nor completely turbulent. The profile is somewhat different at z-position 2, 3, 7 and 8. Where the boundary layer seems to be more laminar at position 3 and 7, it looks fully turbulent at position 2 and 8. This is probably caused by the side of the cavity and the flow structure at the corner. Further away from the side of the cavity, at z-position 1 and 9, the boundary layer profile stays laminar, even at the following strips. Only the profile for z-position 1 at strip 5 seems to be an exception, as can be seen in Figure 6.9. The boundary layer has become fully turbulent at strip 4 and it is even more developed at strip 5 as can be seen in Figure 6.8 and 6.9. These plots are an indication of the expected profiles at the different strips. A more detailed analysis can be obtained from the boundary layer properties. From these results several of these values can be deduced like the thickness of the boundary layer $\delta_{0.99}$, the displacement thickness $\delta^*$, the momentum thickness $\theta$ and the shape factor $H$.

6.3.2 Boundary layer thickness $\delta_{0.99}$

The boundary layer thickness $\delta_{0.99}$ is defined as the distance away from the surface where the local velocity reaches to 99 % of the free stream velocity, that is: $u(y = \delta_{0.99}) = 0.99U_\infty$. The obtained $\delta_{0.99}$ values for each position are presented in Figure 6.10.

**Figure 6.10:** The $\delta$ values at each position for all the strips.

Using the Blasius theory, the boundary layer thickness in the laminar zone is given by:

$$\frac{\delta_{0.99}}{x} = \frac{4.91}{\sqrt{Re_x}}$$

where $Re_x = \frac{U_\infty x}{v}$ represents the Reynolds number.

In the turbulent boundary layer, the boundary thickness is calculated by the formula:

$$\frac{\delta_{0.99}}{x} = \frac{0.37}{\sqrt{Re_x}}$$
Using these formulas, a dimensionless delta value $\Delta$ can be defined:

$$\Delta_{\text{lam}} = \frac{\delta_{\text{lam}}}{\sqrt{\frac{Vx}{U_m}}}$$

$$\Delta_{\text{turb}} = \frac{\delta_{\text{turb}}}{\sqrt[3]{\frac{Vx^4}{U_m}}}$$

The theoretical value for $\Delta_{\text{lam}}$ is 4.91 and for $\Delta_{\text{turb}}$ 0.37. These results are shown in Figure 6.11. For strip 1 and 2 the laminar approach is used, for strip 3, 4 and 5 the turbulent case is applied.

![Graph showing dimensionless $\Delta$ values at each position for all the strips.](image)

Figure 6.11: The dimensionless $\Delta$ values at each position for all the strips.

The values for the laminar boundary layer at strip 1 and 2 are very close (-9%, +7%) to the predicted 4.91. Only the delta for the z-position -235 mm at strip 5 has an extremely high value. This also influences the next parameters at this position. It also explains the transitional boundary layer profile as shown in Figure 6.9. The other points at the two most exterior positions do have a laminar value for all the strips.

The average value for the turbulent boundary layers at strip 3 is 0.52 (-15%, +11%). For strip 4 and 5 this becomes 0.57 (-10%, +7%). This is significantly higher than the expected value 0.37. The flow structure, vortices and instabilities inside the cavity cause the boundary layer to have a larger thickness.

### 6.3.3 Displacement thickness $\delta^*$

The $\delta^*$ values for all the measured positions are presented in Figure 6.12.
Figure 6.12: The $\delta^*$ values at each position for all the strips.

As can be seen in this graph, the pattern is symmetrical. The high values for the $z$-positions just inside the cavity at strip 3, can be related to the vortices that occur on both upstream corners as described by the ball bearings and the fluid as shown in Figure 6.2 c. This different behaviour, especially at the corners and the edges of the cavity, is also noticeable in Figure 6.7, where a different boundary layer profile is shown at $z$-position 3 and 7.

6.3.4 Momentum thickness $\theta$

Figure 6.13 shows the $\theta$ values calculated from the measurements.
The momentum thickness shows the same profile as the displacement thickness. More information can be retrieved by the next parameter, the shape factor H.

6.3.5 Shape factor H
The shape factor is used to characterize the state of the boundary layer:

- \( H = 2.6 \) for an undisturbed, constant-pressure, flat wall laminar boundary layer,
- \( H = 1.3 \) for a fully developed, constant-pressure, flat wall turbulent boundary layer.

Figure 6.14 shows the obtained values for the different strips.

![Figure 6.14: The H values at each position for all the strips.](image)

The average H value for each strip is given in Table 6.1:

<table>
<thead>
<tr>
<th>Strip</th>
<th>Average H</th>
<th>Min, Max</th>
</tr>
</thead>
<tbody>
<tr>
<td>Strip 1</td>
<td>2.58</td>
<td>-3%, +3%</td>
</tr>
<tr>
<td>Strip 2</td>
<td>2.81</td>
<td>-8%, +3%</td>
</tr>
<tr>
<td>Strip 3</td>
<td>1.49</td>
<td>-10%, +21%</td>
</tr>
<tr>
<td>Strip 4</td>
<td>1.34</td>
<td>-4%, +4%</td>
</tr>
<tr>
<td>Strip 5</td>
<td>1.32</td>
<td>-4%, +4%</td>
</tr>
</tbody>
</table>

Table 6.1: The average H values with the accompanying differences.

Strip 1 shows a laminar pattern with small differences. At strip 2, the H value is somewhat higher than the expected 2.6. There still is a laminar boundary layer, but the influence of the cavity - with the re-circulating internal flow and the vortices at the corners - on strip 2 shows from these results. The shape factor for strip 4 and 5 suggests a turbulent boundary layer. For strip 3 however, the value of the shape factor as well as its variations are very high. So just behind the cavity the flow suffers the most from all the effects that take place inside the cavity. These effects weaken further downstream the model.
6.3.6 Pulsation profile

For further investigation of the properties of the boundary layer, also the pulsation velocity profiles are examined. To compare the pulsation velocity profile at each strip, profiles for different span wise z-positions for a strip are plotted in the same picture. The results are shown in Figures 6.15 to 6.19 for each strip. The profiles are given as the dimensionless pulsation intensity $u' = U'/U$ (the pulsation velocity $U'$ divided by the mean velocity $U$) versus the dimensionless height $\eta = y/\delta_{0.99}$ (the height $y$ divided by the thickness of the boundary layer $\delta_{0.99}$).

![Pulsation Intensity Profile](image)

Figure 6.15: Pulsation intensity boundary layer profile for the different positions at strip 1.
Figure 6.16: Pulsation intensity boundary layer profile for the different positions at strip 2.

Figure 6.17: Pulsation intensity boundary layer profile for the different positions at strip 3.
Figure 6.18: Pulsation intensity boundary layer profile for the different positions at strip 4.

Figure 6.19: Pulsation intensity boundary layer profile for the different positions at strip 5.
The average pulsation intensity values $u'$ inside the boundary layer for each strip are presented in Figure 6.20 at all nine z-positions.

![Figure 6.20: The average dimensionless pulsation intensity values at each position for all the strips.](image)

Figure 6.20 shows again a symmetric pattern for the pulsation intensity. Only at z-positions 1, 2 and 3 the values are somewhat higher than the corresponding z-positions 7, 8 and 9 on the other side of the cavity. This is the consequence of another series of measurements with very high pulsation velocities. The average $u'$ value (for positions 4 till 8) for each strip is given in Table 6.2:

<table>
<thead>
<tr>
<th>Strip</th>
<th>$\text{Average } u'$</th>
<th>Min, Max</th>
</tr>
</thead>
<tbody>
<tr>
<td>Strip 1</td>
<td>0.00963</td>
<td>-21%, +26%</td>
</tr>
<tr>
<td>Strip 2</td>
<td>0.03633</td>
<td>-44%, +71%</td>
</tr>
<tr>
<td>Strip 3</td>
<td>0.23911</td>
<td>-25%, +18%</td>
</tr>
<tr>
<td>Strip 4</td>
<td>0.13776</td>
<td>-5%, +3%</td>
</tr>
<tr>
<td>Strip 5</td>
<td>0.07173</td>
<td>-19%, +39%</td>
</tr>
</tbody>
</table>

Table 6.2: The average $u'$ values with the accompanying differences.

The pulsation intensity profile for strip 1 shows a similar profile for all positions as can be seen in Figure 6.15. From Table 6.2, it follows that, as can be expected, the pulsation intensity ($u' = U'/U$) is lowest at strip 1. At strip 2, shown in Figure 6.16, all positions have a different value for $u'$. The average value is also a lot higher than at strip 1. This is due to the internal recirculating flow, which also has an influence on the shape factor discussed in section 6.3.5. Z-positions 7 and 8 show again a different behaviour in Figure 6.17 at strip 3. Also the mean velocity profile in Figure 6.7 and the boundary layer properties present different values at these positions. At strip 4 and strip 5 however, all profiles have the same values. Position 9 is again an exception, but at that point there is no influence from the cavity. The pulsation intensity reaches a maximum just behind the cavity at strip 3. The turbulent boundary layer becomes more stable after the cavity, so the pulsation intensity decreases as shown in Table 6.2 for strip 4 and 5.
7. Numerical results

7.1 Computational studies

Before the experimental results are compared with the numerical values, a summary of some computational results is given. The study of unsteady incompressible flows over 3-D shallow rectangular cavities has been performed by Yao [48,49] at the Queen’s University of Belfast. The cavity has a length-to-depth ratio $D/L = 0.25$ and a width-to-length ratio $W/L = 3$. A laminar Blasius boundary layer is taken as the inflow condition, so the same conditions as for the experiment were used.

Cavity flow fields are first calculated for a Reynolds number of 5,000 and 10,000 [48]. This was done using a CFD approach, based on the unsteady Navier-Stokes equations for 3-dimensional incompressible flow. The instantaneous velocity vectors of the longitudinal $x$-direction at the $z = 0.0$ and $z = 0.8$ plane are shown in Figure 7.1. The coordinate $z = 0$ corresponds with the middle of the cavity, $z = 1$ corresponds with the side of the cavity.

Figure 7.1: Instantaneous flow velocity vectors of $x$-direction at $z=0$ and $z=0.8$ planes for $Re = 5,000$ (a) and $Re = 10,000$ (b) [48].
A primary vortex at the downstream side of the cavity and a secondary vortex upstream can be seen in each picture. These vortices are stronger at Re = 10,000 and also the side of the cavity has an influence on the strength of the vortices. Figure 7.1 also shows the Kelvin-Helmholtz instability in the shear layer. The development of this Kelvin-Helmholtz instability as well as the unsteady structure of the cavity flow is shown in Figure 7.2.

Figure 7.2: Flow structures at the z = 0.3 plane at t = 40, 45, 50 and 55 with Re = 10,000 [48].

The instantaneous cross flow velocity vectors are shown in Figure 7.3 for Re = 5,000 and Re = 10,000. Three lateral planes are presented for one half of the cavity based on symmetry, two inside the cavity (x = 0.6 and x = 0.85) and one downstream the cavity (x = 1.7). The coordinate x = 0 corresponds with the front side of the cavity, x = 1 corresponds with the downstream side of the cavity.
At the $x = 0.6$ plane, longitudinal vortices at the top of the cavity are shown for both Reynolds numbers. For the $x = 0.85$ plane, the longitudinal vortices were found at the floor of the cavity. The flow structure has become more complex, especially at the side of the cavity. Downstream the cavity, at $x = 1.7$, the longitudinal vortices become even stronger. For $Re = 10,000$, the flow structure is more complex than for $Re = 5,000$. So as the Reynolds number increases, the cavity flow fields become highly unsteady and complex and show a 3-dimensional pattern.

Using a large eddy simulation approach, the following additional results were found for $Re = 5,000$ and $Re = 50,000$ [49]. Figure 7.4 shows the contours of the streamwise velocity and the spanwise vorticity for $Re = 5,000$.

The streamwise velocity picture 7.4 (a) shows a weak but detectable crossflow. Figure 7.4 (b) shows the Kelvin-Helmholtz instability as noted by Figure 7.1. For $Re = 5,000$, the flow fields are disturbed by the cavity, resulting in an unsteady and complex flow, but the flow does not
become turbulent. For higher Reynolds numbers, for example \( \text{Re} = 50,000 \), the laminar to turbulent transition can occur. This is demonstrated in Figure 7.5, where a time sequence of streamwise velocity contours in the horizontal plane at \( y/\delta^* = 0.26 \) is shown.

![Figure 7.5: Contours of the streamwise velocity in the horizontal plane at \( y/\delta^* = 0.26 \), for \( \text{Re}=50,000 \) [49].](image)

At \( t = 8 \), Figure 7.5 (a), only small vortices are seen around the side walls of the cavity. At \( t = 12 \), \( \Lambda \)-vortices appear in the shear layer. They are already strong enough to go in transition as shown in Figure 7.5 (b), so laminar breakdown occurs. At \( t = 16 \), the laminar to turbulent process continues. The laminar to turbulent transition can be described from Figure 7.5 by three regions: upstream of, above and downstream the cavity. Upstream of the cavity, 2-dimensional Tollmien-Schlichting waves are present in the boundary layer. Above the cavity, 3-dimensional disturbances occur in the shear layer. As a consequence, transition occurs and the laminar flow is approaching breakdown. Downstream the cavity, the flow becomes turbulent.

### 7.2 Comparison theoretical and experimental data

In this section, the boundary layer profiles discussed in chapter six (section 6.3.1) are compared with the numerical CFD results obtained by Yao [50] for \( \text{Re} = 100,000 \). For each strip, the \( z \)-positions 4 till 9 are shown in Figures 7.6 to 7.10. The profiles are plotted as the dimensionless velocity \( u = U/U_\infty \) (the velocity \( U \) divided by the maximum velocity \( U_\infty \)) versus the height \( y \). The symbols indicate the experimental data and the lines represent the numerical results.
a) dimensionless velocity profile for z-positions 4, 5 and 6
b) dimensionless velocity profile for z-positions 7, 8 and 9

Figure 7.6: The stream wise velocities at strip 1 at different positions compared with the numerical results (numerical results: lines, experimental data: symbols) [50].
Figure 7.7: The stream wise velocities at strip 2 at different positions compared with the numerical results (numerical results: lines, experimental data: symbols) [50].
Figure 7.8: The stream wise velocities at strip 3 at different positions compared with the numerical results (numerical results: lines, experimental data: symbols) [50].
Figure 7.9: The streamwise velocities at strip 4 at different positions compared with the numerical results (numerical results: lines, experimental data: symbols) [50].
Figure 7.10: The streamwise velocities at strip 5 at different positions compared with the numerical results (numerical results: lines, experimental data: symbols) [50].
Upstream of the cavity, the boundary layer thickness of the experimental data is thinner than the CFD model boundary layer. The experimental data at strip 1 and 2 differ thus from the numerical values as shown in Figure 7.6 and 7.7. Nevertheless, all profiles show a laminar behaviour. Downstream the cavity, good agreement can be found between experimental and computational results. At position 9, the boundary layer stays laminar for both the experimental and the CFD values. The cavity has thus no influence on the flow at this position. In Figure 7.8, just downstream the cavity, a large difference can be seen at position 7, where vortices and vertical flow structures are generated by the cavity corner. Further downstream the cavity, the experimental results are more similar to the numerical data. The prediction of laminar to turbulent transition by the computational studies is thus confirmed by the experimental results. The laminar profiles at strips 1 and 2 become turbulent boundary layer profiles at strips 4 and 5. A small remark on this topic has to be made. From the experiments, only time-averaged results are available for the time-varying flow characteristics. These results are different from the computational results based on time-dependent flow models.

Another parameter that can be compared, is the value of the shape factor \( H \). In figure 7.11, the numerical \( H \) values at the middle of the cavity are shown for a Reynolds number \( \text{Re} = 5,000 \) and \( \text{Re} = 50,000 \) [49]. Also the measured \( H \) values for the middle of the cavity are presented.

![Figure 7.11: Streamwise variation of shape factor for numerical (a,b) [49] and experimental (c) data with different Reynolds numbers at the middle of the cavity.](image-url)
Figure 7.11 (a) shows a weakly nonlinear growth of the disturbance after the cavity, but the flow stays laminar for Re = 5,000. For Re = 50,000, a laminar breakdown of the flow is observed and where after transition to turbulence occurs. The profile for the experimental H values, Figure 7.11 (c), is also laminar upstream of the cavity and downstream the cavity, a turbulent boundary layer is observed for Re = 100,000.
8. Conclusion

The two visualisation methods, the miniature balls and the light-reflecting liquid, show the flow pattern inside the cavity. From these results, a large downstream vortex over the whole width of the cavity and two vortices at the upstream corners of the cavity are observed. This is shown in Figure 8.1. The downstream or primary vortex was also observed by Neary and Stephanoff [24] as discussed in chapter 3 and shown in Figure 8.2 (a). Numerical CFD computations, performed by Hao [48] and discussed in chapter 7, predict this primary vortex (Figure 8.2 (b)) and the flow behaviour at the corners as shown in Figure 8.3.

![Figure 8.1](image1)

Figure 8.1: (a) The streamwise flow pattern over the open cavity and (b) a top view of the cavity with the vortices at the upstream corners.

![Figure 8.2](image2)

Figure 8.2: a) The fluid motion in the cavity with three vortices from Neary and Stephanoff [24] (a) and numerical calculations from Hao [48] (b).

![Figure 8.3](image3)

Figure 8.3: Instantaneous cross-flow velocity vectors for one half of the cavity at the lateral plane x=0.85 (relative to the front wall of the cavity) for Re=10,000 [48].

The boundary layer profiles, obtained from the hot-wire velocity measurements, show that the boundary layer stays laminar in front of the cavity. Just downstream the cavity, a transitional profile occurs. The influence of the downstream cavity corners can be seen in these profiles; a different pattern can be observed for different spanwise positions. Further downstream the cavity, the boundary layer profiles become completely turbulent. The influence of the cavity corners decreases as the profiles all show the same pattern. The cavity does not influence the
whole flow profile across the model. At a certain distance from the side of the cavity, the boundary layer profile stays laminar, even downstream the cavity.

From the hot-wire results several values can be deduced like the thickness of the boundary layer $\delta_{0.99}$, the displacement thickness $\delta^*$, the momentum thickness $\theta$ and the shape factor $H$. The boundary layer values $\delta_{0.99}$ upstream the cavity have a laminar value as well as these at the side of the cavity where a laminar boundary layer was observed. Downstream the cavity, a higher $\delta_{0.99}$ value than for turbulent boundary layers is found. The flow structure, vortices and instabilities inside the cavity cause the boundary layer to have a larger thickness.

A symmetrical pattern around the cavity is observed for the displacement thickness $\delta^*$ and the momentum thickness $\theta$. Extremely high values are observed at the corners just downstream the cavity. This is again due to the vortices and instabilities at the corners and it is related to the boundary layer profile at these positions.

The shape factor $H$, which is defined as the ratio of the displacement to the momentum thickness, shows a laminar value before the cavity and a turbulent value downstream the cavity. Just before and after the cavity, however, these values are a little bit higher. This is due to the recirculating internal flow and the vortices at the corners. Just behind the cavity the flow suffers the most from all the effects that take place inside the cavity. These effects weaken further downstream the model.

A similar analysis can be made for the pulsation profiles and the average pulsation intensities. Just before and after the cavity, different pulsation intensity profiles are observed for the spanwise positions. Also the average pulsation intensities at these positions are influenced by the same effects inside the cavity as discussed earlier. The pulsation intensity reaches a maximum just behind the cavity. Further downstream, the turbulent boundary layer becomes more stable after the cavity, so the average pulsation intensity decreases. At a certain distance from the side of the cavity, also the pulsation intensity profile stays laminar.

The experimental boundary layer profiles were also compared with the numerical CFD results obtained by Yao [50]. Upstream of the cavity, the boundary layer thickness of the experimental data is thinner than the CFD model boundary layer. The experimental data differ thus from the numerical values. Downstream the cavity, good agreement is found between experimental and computational results. Only at the corners of the cavity some differences can be observed. Also the numerical data predict that at a certain distance from the side of the cavity, the profile stays laminar, even downstream the cavity.

The prediction of laminar-to-turbulent transition by the computational studies is thus confirmed by the experimental results. The laminar profiles in front of the cavity become turbulent boundary layer profiles downstream the cavity as shown by the parameters discussed above. A detailed analysis of the pulsation intensities, however, would be useful to better understand the transition processes to a turbulent flow. The power spectra of these pulsations can give more information about the frequencies of Tollmien-Schlichting waves and Kelvin-Helmholtz instabilities, which occur in the laminar-to-turbulent transition. Some of these spectra are presented in Appendix H.
Appendix A: Equations for incompressible Newtonian fluids

- The equation for the conservation of mass is given by:

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0, \]  
(A1)

which is also called the continuity equation. This can also be written as:

\[ \frac{D\rho}{Dt} + \rho \nabla \cdot \vec{v} = 0, \]  
(A2)

with: \( \frac{D\rho}{Dt} = \frac{\partial \rho}{\partial t} + \vec{v} \cdot \nabla \rho \). For an incompressible fluid follows \( \frac{D\rho}{Dt} = 0 \) so equation A2 becomes:

\[ \nabla \cdot \vec{v} = 0. \]  
(A3)

- The general momentum conservation equation is given by:

\[ \frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \vec{v}) = \rho \vec{g} + \nabla \cdot \underline{S}, \]  
(A4)

with \( \rho \) the fluid density, \( t \) the time, \( \vec{v} \) the velocity vector, \( \vec{g} \) the gravitational acceleration and \( \underline{S} \) the stress tensor (also called a stress matrix with nine stress components). This equation can be rewritten using the following rules:

\[ \frac{\partial (\rho \vec{v})}{\partial t} = \vec{v} \frac{\partial \rho}{\partial t} + \rho \frac{\partial \vec{v}}{\partial t}, \]  
(A5)

and

\[ \nabla \cdot (\rho \vec{v} \vec{v}) = \vec{v} \nabla \cdot (\rho \vec{v}) + \rho (\vec{v} \cdot \nabla) \vec{v}. \]  
(A6)

Based on the continuity equation A1, the sum of the first terms on the right side of A5 and A6 turns out to be zero. So A4 becomes:

\[ \rho \left[ \frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla) \vec{v} \right] = \rho \vec{g} + \nabla \underline{S}. \]  
(A7)

The stress tensor \( \underline{S} \) can be replaced by:

\[ \underline{S} = -p \underline{I} + \underline{\sigma}, \]  
(A8)

with \( p \) the pressure and \( \underline{\sigma} = 0 \) in case of a uniform velocity flow \( \vec{v} \).

Equation A7 can thus be written as:
\[ \rho \left[ \frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla) \vec{v} \right] = \rho \vec{g} - \nabla p + \nabla \cdot \sigma. \quad (A9) \]

\( \sigma \) is related to the strain-rate tensor \( D \) and the dilatation velocity \( \nabla . \vec{v} \). One can say that:

1. \( \sigma = 0 \), if \( \nabla . \vec{v} = 0 \),
2. \( \sigma \) is a linear function of \( D \) and \( \nabla . \vec{v} \),
3. the medium is isotropic, so the fluid properties are invariant to rotations.

So it follows that:

\[ \sigma = 2\mu \left[ \frac{D}{3} - \frac{1}{3} (\nabla \cdot \vec{v}) \mathbb{I} \right] + \zeta (\nabla \cdot \vec{v}) \mathbb{I}, \quad (A10) \]

with \( \mu \) the shear viscosity and \( \zeta \) the bulk viscosity. The first term on the right side of A10 is related to deformation, the second term is related to dilatation. A fluid that satisfies A10 is called a Newtonian medium.

Inserting the expression A10 for \( \sigma \) in A9 results in:

\[ \rho \left[ \frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla) \vec{v} \right] = \rho \vec{g} - \nabla p + \nabla \cdot (2\mu D) + \nabla \left[ (\zeta - \frac{2}{3} \mu)(\nabla \cdot \vec{v}) \right]. \quad (A11) \]

The viscosity components \( \mu \) and \( \zeta \) are treated as constants. The strain-rate tensor \( D \) can be written as:

\[ D_{ij} = \frac{1}{2} \left( \frac{\partial \vec{v}_i}{\partial x_j} + \frac{\partial \vec{v}_j}{\partial x_i} \right). \quad (A12) \]

For the derivative follows then:

\[ \frac{\partial D_{ij}}{\partial x_k} = \frac{1}{2} \left( \frac{\partial}{\partial x_k} \left( \frac{\partial \vec{v}_i}{\partial x_j} \right) + \frac{\partial^2 \vec{v}_j}{\partial x_i \partial x_k} \right) = \frac{1}{2} \frac{\partial}{\partial x_j} (\nabla \cdot \vec{v}) + \frac{1}{2} \nabla^2 \vec{v}_j. \quad (A13) \]

Inserting this in equation A11 gives:

\[ \rho \left[ \frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla) \vec{v} \right] = \rho \vec{g} - \nabla p + \mu \nabla^2 \vec{v} + (\zeta + \frac{1}{3} \mu) \nabla (\nabla \cdot \vec{v}). \quad (A14) \]

For an incompressible flow the continuity equation \( \nabla . \vec{v} = 0 \) (A3) can be used so:

\[ \rho \left[ \frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla) \vec{v} \right] = \rho \vec{g} - \nabla p + \mu \nabla^2 \vec{v}, \quad (A15) \]
which is known as the Navier-Stokes equation.

Appendix B: The Blasius profile

In this appendix, the Blasius solution for a boundary layer over a semi-infinite flat plate in a parallel flow with constant velocity \( U_\infty \) is deduced. For a flat plate, it follows that \( \frac{dU_\infty}{dx} = 0 \) and, as a consequence, \( \frac{dp}{dx} = 0 \). The Prandtl equations A3 and A15 then reduce to:

- \( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0, \) \hspace{1cm} (B1)
- \( u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \nu \frac{\partial^2 u}{\partial y^2}, \) \hspace{1cm} (B2)

with the boundary conditions:

- \( u = v = 0 \) for \( y = 0 \),
- \( u \to U_\infty \) for \( y \to \infty \).

This problem is solved in 1908 by Blasius as follows. Because there is no external enforced longitudinal scale in the x-direction, the solutions on the different x-positions are necessarily uniform. As a consequence, the different velocity profiles can be pictured on each other, which implies that:

\[ \frac{u}{U_\infty} = g(\xi), \] \hspace{1cm} (B3)

with \( g(\xi) \) a function of the (dimensionless) uniformity co-ordinate

\[ \xi = \frac{y}{\delta(x)}. \] \hspace{1cm} (B4)

Then the stream function \( \psi \) is defined as:

\[ u = \frac{\partial \psi}{\partial y}, v = -\frac{\partial \psi}{\partial x}. \] \hspace{1cm} (B5)

Based on the relation B3, \( \psi \) can be formulated as:

\[ \psi = \int_0^y u dy = \delta \int_0^\xi u d\xi = \delta \int_0^\xi U_\infty g(\xi) d\xi = U_\infty \delta f(\xi), \] \hspace{1cm} (B6)

with

\[ g(\xi) = \frac{df}{d\xi}. \] \hspace{1cm} (B7)

The function \( f(\xi) \) can be considered as the dimensionless stream function following from B6:

\[ f(\xi) = \frac{\psi}{U_\infty \delta}. \] \hspace{1cm} (B8)
The stream function \( \psi \) is thus scaled with the local (2D) volumeflux \( U_\infty \delta \). In view of the momentum equation B2, the following parameters are rewritten in terms of \( f \) and \( \xi \):

\[
\begin{align*}
u &= -\frac{\partial \psi}{\partial x} = -U_\infty f \frac{d\delta}{dx} - U_\infty \delta f' \frac{\partial \xi}{\partial x} = -U_\infty \frac{d\delta}{dx} (f - \xi f') , \\
u &= -\frac{\partial \psi}{\partial y} = -\frac{\partial \psi}{\partial \xi} \frac{\partial \xi}{\partial y} = -\frac{\partial \psi}{\partial \xi} \frac{\partial \xi}{\partial y} - \frac{\partial \psi}{\partial \xi} \frac{\partial \xi}{\partial y} = -U_\infty \frac{\partial \xi}{\partial \xi} \frac{d\delta}{dx} , \\
\frac{\partial u}{\partial x} &= \frac{\partial u}{\partial \xi} \frac{\partial \xi}{\partial x} = U_\infty \frac{d\xi}{dx} \left( \frac{y}{\delta} \right) = -U_\infty \frac{d\xi}{dx} f'' , \\
\frac{\partial u}{\partial y} &= \frac{\partial u}{\partial \xi} \frac{\partial \xi}{\partial y} = U_\infty \frac{d\xi}{dx} f'' , \\
\frac{\partial^2 u}{\partial y^2} &= \frac{\partial}{\partial \xi} \left( \frac{\partial u}{\partial y} \frac{\partial \xi}{\partial y} \right) = \frac{U_\infty}{\delta^2} f''' ,
\end{align*}
\]

with the derivatives to \( \xi \). Substitution in B2 results in:

\[
-\left( \frac{U_\infty \delta}{v} \frac{d\delta}{dx} \right) f'' = f'' .
\]

The function \( f \) and its derivatives are not explicitly dependent of \( x \), which implies that:

\[
\frac{U_\infty \delta}{v} \frac{d\delta}{dx} = \text{cte} .
\]

The constant is given the value \( \gamma_* \), so after integration follows:

\[
\delta = \sqrt{\frac{v x}{U_\infty}} .
\]

The original problem can now be reformulated as:

\[
2 \frac{d^3 f}{d\xi^3} + f \frac{d^2 f}{d\xi^2} = 0 ,
\]

with the boundary conditions written in terms of \( f \):

\[
\begin{align*}
f &= \frac{df}{d\xi} = 0 \quad \text{for } \xi = 0 , \\
\frac{df}{d\xi} &\to 1 \quad \text{for } \xi \to \infty .
\end{align*}
\]
So the non-linear partial differential equation B2 is transformed to an (also non-linear) normal differential equation. Blasius found a solution for this equation by series expansion. The velocity distribution \( \frac{u}{U_\infty} = f' (\xi) \) of this numerical solution is presented in Figure 3.2. This profile has an inflexion point at the wall \( (\xi = 0) \), where \( \frac{\partial^2 u}{\partial y^2} = 0 \). This is due to the absence of the pressure gradient \( \frac{\partial p}{\partial x} \).

**Appendix C: Pohlhausen’s method for a laminar boundary layer**

To find the velocity profile for a laminar boundary layer, Pohlhausen assumed a fourth degree polynomial of the form:

\[
\frac{u(x, y)}{U(x, 0)} = A(x) \left( \frac{y}{\delta} \right) + B(x) \left( \frac{y}{\delta} \right)^2 + C(x) \left( \frac{y}{\delta} \right)^3 + D(x) \left( \frac{y}{\delta} \right)^4, \tag{C1}
\]

as a solution. The boundary conditions are:

- \( y = 0 : u = v = 0 \Rightarrow \mu \frac{\partial^2 u}{\partial y^2} = \frac{\partial p}{\partial x}, \tag{C2} \)
- \( y = \delta : u = U \Rightarrow \frac{u}{U} = 1, \frac{\partial u}{\partial y} = 0, \frac{\partial^2 u}{\partial y^2} = 0 \). \tag{C3} \)

For \( \frac{y}{\delta} = 1 \) follows thus from C1 and the boundary condition C3:

1. \( A + B + C + D = 1, \tag{C4} \)
2. \( \frac{1}{U} \frac{\partial u}{\partial y} = 0 = A + 2B + 3C + 4D, \tag{C5} \)
3. \( \frac{1}{U} \frac{\partial^2 u}{\partial y^2} = 0 = 2B + 6C + 12D. \tag{C6} \)

For \( y = 0 \), follows from C2:

\[
\frac{1}{U} \frac{\partial^2 u}{\partial y^2} = \frac{2B}{\delta^2} + \frac{6C}{\delta^3} \frac{y}{\delta} + \frac{12D}{\delta^4} y^2 = \frac{2B}{\delta^2} \Rightarrow B = \frac{\delta^2 \frac{\partial p}{\partial x}}{2 \mu U \frac{\partial x}{\partial x}} \tag{C7} \)

The latter equation can be rewritten using the Bernoulli equation:

\[
\frac{\partial p}{\partial x} = -\rho U \frac{\partial U}{\partial x}, \tag{C8} \)

to:
\[ B = -\frac{\delta^2}{2} \frac{\partial U}{\partial x} = \frac{-\lambda}{2}, \] (C9)

with \( \lambda = \frac{\delta^2}{2} \frac{\partial U}{\partial x} \) the Pohlhausen’s parameter. Now a value for \( B \) has been obtained, the other parameters \( A, C \) and \( D \) can also be determined from equations C4 to C6:

\[ A = 2 + \frac{\lambda}{6}, \] (C10)
\[ C = -2 + \frac{\lambda}{2}, \] (C11)
\[ D = 1 - \frac{\lambda}{6}. \] (C12)

Inserting these values in the fourth-order polynomial C1 results in:

\[ \frac{u}{U} = 2\frac{y^2}{\delta} - \frac{y^3}{\delta} + \frac{1}{2} \frac{y^4}{\delta} + \frac{\lambda}{6} \frac{y}{\delta} - 3 \frac{y^2}{\delta} + 3 \frac{y^3}{\delta} - \frac{y^4}{\delta}. \] (C13)

The Pohlhausen’s parameter \( \lambda \) contains the pressure gradient effect, so for a semi-infinite flat plate in a parallel flow with constant velocity \( U_\infty \) and a zero pressure gradient, equation C13 limits to:

\[ \frac{u}{U_\infty} = 2\frac{y^2}{\delta} - \frac{y^3}{\delta} + \frac{1}{2} \frac{y^4}{\delta}. \] (C14)

**Appendix D: Von Karman momentum equation**

In general it’s not possible to find an exact solution of the boundary layer equations A3 and A15. Therefore, Von Karman integrated the x-component of the momentum equation over the boundary layer thickness as follows.

The x-component of the momentum equation is given by:

\[ u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = U_\infty \frac{dU_\infty}{dx} + v \frac{\partial^2 u}{\partial y^2}. \] (D1)

The factor \( u \frac{dU_\infty}{dx} \) is added and subtracted from D1 and so this can be rewritten to:

\[ (U_\infty - u) \frac{dU_\infty}{dx} + u \frac{\partial}{\partial x} (U_\infty - u) + v \frac{\partial}{\partial y} (U_\infty - u) = -v \frac{\partial^2 u}{\partial y^2}. \] (D2)

This equation is integrated in the y-direction from the wall (\( y = 0 \)) till a point outside the boundary layer (\( y = h, \) with \( h > \delta \)). For the first term this results in:
Partial integration of the third term gives:
\[
\int_0^h \frac{\partial}{\partial y} (U_\infty - u) \, dy = [v(U_\infty - u)]_0^h - \int_0^h \frac{\partial v}{\partial y} (U_\infty - u) \, dy = \int_0^h \frac{\partial u}{\partial x} (U_\infty - u) \, dy,
\]
(D4)

based on: \( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \),
\[ v(y = 0) = 0, \quad v(y = h) = U_\infty. \]

The term on the right side of D2 gives after integration:
\[
-\nu \int_0^h \frac{\partial^2 u}{\partial y^2} \, dy = -v \frac{\partial u}{\partial y} \bigg|_0^h = \frac{\tau_0}{\rho},
\]
(D5)

with: \( \tau_0 = \mu \frac{\partial u}{\partial y} \bigg|_{y=0} \) the shear stress at the wall.

So the integrated equation D2 becomes:
\[
U_\infty \delta^* \frac{dU_\infty}{dx} + \int_0^h \left[ u \frac{\partial}{\partial x} (U_\infty - u) + (U_\infty - u) \frac{\partial u}{\partial x} \right] \, dy = \frac{\tau_0}{\rho}.
\]
(D6)

The integral can be solved as follows:
\[
\int_0^h \frac{\partial}{\partial x} [u(U_\infty - u)] \, dy = \frac{d}{dx} \int_0^h u(U_\infty - u) \, dy = \frac{d}{dx} (U_\infty^2 \theta).
\]
(D7)

So D6 can be written as:
\[
\frac{d(\theta U_\infty^2)}{dx} + \delta^* U_\infty \frac{dU_\infty}{dx} = \frac{\tau_0}{\rho}.
\]
(D8)

This result is known as the Von Karman integral momentum equation.
Appendix E: Turbulent boundary layer theory

Based on experimental results, the velocity profile for a large range of Re-values is described by:

\[ \frac{\bar{U}}{u} = 8.7(y^*)^{1/n}, \quad (E1) \]

with: \( u^* = \left( \frac{\tau_0}{\rho} \right)^{1/2} \) the friction velocity at the wall,

\[ y^* = \frac{yu}{v} \] the dimensionless height coefficient.

At the edge of the boundary layer, \( y = \delta \), the velocity \( \bar{U} \) should be the mean stream velocity \( U_\infty \), so:

\[ \frac{U_\infty}{u^*} = 8.7 \left( \frac{\delta}{\nu} \right)^{1/n}. \quad (E2) \]

From this relation, \( u^* \) and \( \tau_0 = \rho(u^*)^2 \) can be determined in function of the boundary layer thickness \( \delta \). It follows that:

\[ (u^*/U_\infty)^2 = (8.7)^{-2n/(n+1)} \left[ \frac{\nu}{\delta U_\infty} \right]^{2/(n+1)}. \quad (E3) \]

The quotient of these equations E1 and E2 results in:

\[ \frac{\bar{u}}{U_\infty} = \left( \frac{y}{\delta} \right)^{1/n}. \quad (E4) \]

For \( n = 7 \), this equation is also called the Prandtl one-seventh power law for a turbulent boundary layer over a flat plate. With E4 the displacement thickness \( \delta^* \) and the momentum thickness \( \theta \) can be calculated:

\[ \delta^* = \int_0^\infty (1 - \bar{U}/U_\infty) dy = \delta/(n+1), \quad (E5) \]

\[ \theta = \int_0^\infty (\bar{U}/U_\infty)(1 - \bar{U}/U_\infty) dy = \delta \frac{n}{(n+1)(n+2)}. \quad (E6) \]

The results E3, E5 and E6 can be used in the Von Karman momentum equation (D8):

\[ \frac{d(\theta U_\infty^2)}{dx} + \delta^* U_\infty \frac{dU_\infty}{dx} = (u^*)^2. \quad (E7) \]

In case of a flat plate, with \( \frac{dU_\infty}{dx} = 0 \), follows thus for E7:
Integration of this equation gives:

\[ \frac{\delta}{x} = A_n \text{Re}_x^{-2/(n+3)}, \]  

with: 

\[ \text{Re}_x = \frac{x U_{\infty}}{v}, \]

\[ A_n = \left[ \frac{(n+3)(n+2)}{n} \right]^{-2/(n+3)} \cdot (8.7)^{-2n/(n+3)}. \]

For \( n = 7 \), equation E9 becomes:

\[ \frac{\delta}{x} = 0.37 \text{Re}_x^{-1/5}, \]

which gives the boundary layer thickness for a turbulent boundary layer over a flat plate.

**Appendix F: Derivation of the Orr-Sommerfeld equation**

The motion of a flow consists of a basic flow and a superimposed perturbation motion. The Cartesian velocity components for the basic flow are given by \( U, V \) and \( W \), the pressure is represented by \( P \). The time varying disturbances, which are small compared to the basic flow quantities, are described by \( u', v', w' \) and \( p' \). The total velocity and pressure are thus given by:

\[ u = U + u', \quad v = V + v', \quad w = W + w', \]  

(N1)

\[ p = P + p'. \]  

(F2)

A 2-dimensional incompressible basic boundary layer flow is described by:

\[ U(y), \quad V = W = 0, \quad P(x,y). \]  

(F3)

The 2-dimensional perturbations are given by:

\[ u'(x,y,t), \quad v'(x,y,t), \quad p'(x,y,t), \]  

(F4)

resulting in the motion according to F1 and F2:

\[ u = U + u', \quad v = v', \quad w = 0, \quad p = P + p'. \]  

(F5)

The basic flow F3 is assumed to be a solution of the Navier-Stokes equations, but also the resulting motion F5 has to satisfy the Navier-Stokes equations. Inserting F5 into the NS-equations, the following equations are obtained (the quadratic terms for the perturbation velocities are ignored):

\[ \frac{\partial u'}{\partial t} + U \frac{\partial u'}{\partial x} + v \frac{\partial u'}{\partial y} + \frac{1}{\rho} \frac{\partial P}{\partial x} + \frac{1}{\rho} \frac{\partial p'}{\partial x} = \nu \left[ \frac{d^2 U}{dy^2} + \Delta u' \right], \]  

(F6)

\[ \frac{\partial v'}{\partial t} + U \frac{\partial v'}{\partial x} + v \frac{\partial v'}{\partial y} + \frac{1}{\rho} \frac{\partial P}{\partial y} + \frac{1}{\rho} \frac{\partial p'}{\partial y} = \nu \Delta v', \]  

(F7)
\[
\frac{\partial u'}{\partial x} + \frac{\partial v'}{\partial y} = 0,
\]
with \(\Delta\) the operator \(\frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2}\).

The basic flow for a boundary layer satisfies the NS-equations, thus these equations can be simplified to:

\[
\begin{align*}
\frac{\partial u'}{\partial t} + U \frac{\partial u'}{\partial x} + v \frac{\partial U}{\partial y} + \frac{1}{\rho} \frac{\partial p'}{\partial x} &= \nu \Delta u', \\
\frac{\partial v'}{\partial t} + U \frac{\partial v'}{\partial x} + \frac{1}{\rho} \frac{\partial p'}{\partial y} &= \nu \Delta v', \\
\frac{\partial u'}{\partial x} + \frac{\partial v'}{\partial y} &= 0.
\end{align*}
\]

The pressure \(p'\) can be eliminated from F9 and F10. Together with F11, two equations are so obtained for \(u'\) and \(v'\) with the boundary no-slip condition: \(u' = v' = 0\).

Assume now a basic flow in the \(x\)-direction with velocity \(U(y)\) with a superimposed perturbation made up of single partial perturbations or modes. Each of these modes is a wave propagating in the \(x\)-direction. A stream function \(\psi\) can be introduced for the two-dimensional perturbation such that:

\[
\begin{align*}
\frac{\partial u'}{\partial y} &= \frac{\partial \psi}{\partial x}, \\
\frac{\partial v'}{\partial x} &= -\frac{\partial \psi}{\partial y}.
\end{align*}
\]

The following stream function of one mode in the perturbation is taken as a trial solution:

\[
\psi(x, y, t) = \varphi(y) e^{i(\alpha x - \beta t)}.
\]

Any disturbance can be described by these Fourier modes. The wavelength of the disturbance \(\lambda = 2\pi/\alpha\) is assumed to be real. The quantity \(\beta\) is complex:

\[
\beta = \beta_r + i \beta_i,
\]
with \(\beta_r\) the frequency of the mode, 
\(\beta_i\) the amplification factor.

The components of the perturbation velocity can now be calculated:

\[
\begin{align*}
u' &= \frac{\partial \psi}{\partial y} = \varphi'(y) e^{i(\alpha x - \beta t)}, \\
v' &= -\frac{\partial \psi}{\partial x} = -i\alpha \varphi(y) e^{i(\alpha x - \beta t)}.
\end{align*}
\]
With these values, the equations F9 and F10 can be rewritten. Therefore, the derivative of these equations gives:

\[
\frac{\partial^2 u'}{\partial y \partial t} + \frac{dU}{dy} \frac{\partial u'}{\partial x} + \frac{\partial^2 u'}{\partial y^2} + \frac{\partial v'}{\partial x} \frac{dU}{dy} + v' \frac{d^2 U}{dy^2} + \frac{\partial^2 p'}{\rho \partial y \partial x} = \frac{\partial}{\partial y} (v \Delta u'), \tag{F17}
\]

\[
\frac{\partial^2 v'}{\partial x \partial t} + \frac{\partial U}{\partial x} \frac{\partial v'}{\partial x} + \frac{\partial^2 v'}{\partial x^2} + \frac{\partial U}{\partial x} \frac{\partial v'}{\partial x} - \frac{\partial^2 p'}{\rho \partial x \partial y} = \frac{\partial}{\partial x} (v \Delta v'). \tag{F18}
\]

Substraction of F18 from F17 results in:

\[
\frac{\partial^2 u'}{\partial y \partial t} + \frac{dU}{dy} \frac{\partial u'}{\partial x} + \frac{\partial^2 u'}{\partial y^2} + \frac{\partial v'}{\partial x} \frac{dU}{dy} + v' \frac{d^2 U}{dy^2} - \frac{\partial^2 v'}{\partial x \partial t} - \frac{\partial U}{\partial x} \frac{\partial v'}{\partial x} - \frac{\partial^2 p'}{\rho \partial x \partial y} = \frac{\partial}{\partial y} (v \Delta u') - \frac{\partial}{\partial x} (v \Delta v'). \tag{F19}
\]

Inserting the values F15 and F16 for \(u'\) and \(v'\) gives for the left side of F19:

\[
-\beta \phi''(y) + i \alpha U' \phi'(y) + i \alpha U \phi''(y) - i \alpha \phi'(y)U' - i \alpha \phi(y)U'' + i \alpha^2 \beta \phi(y) - i \alpha^3 \phi(y)U \tag{F20}
\]

The terms in F20 can be grouped:

\[
U \left( -i \alpha \phi''(y) - i \alpha^2 \phi(y) \right) - i \beta \phi''(y) + i \alpha^2 \beta \phi(y) - i \alpha U'' \phi(y)
= i \alpha U \left( \phi''(y) - \alpha^2 \phi(y) \right) - i \beta \left( \phi''(y) - \alpha^2 \phi(y) \right) - i \alpha U'' \phi(y)
= i \alpha \left[ \left( U - \frac{\beta}{\alpha} \right) \left( \phi''(y) - \alpha^2 \phi(y) \right) - U'' \phi(y) \right], \tag{F21}
\]

with \(c = \frac{\beta}{\alpha}\).

The right side of F19 becomes:

\[
\nu \left[ -\alpha^2 \phi''(y) + \phi'''(y) \right] - \nu \left[ -\alpha^4 \phi(y) + \alpha^2 \phi''(y) \right]
= \nu \left[ -\alpha^2 \phi''(y) + \phi'''(y) + \alpha^4 \phi(y) - \alpha^2 \phi''(y) \right]
= \nu \left[ \phi'''(y) - 2 \alpha^2 \phi''(y) + \alpha^4 \phi(y) \right]. \tag{F22}
\]

So F21 and F22 are used to rewrite expression F19:

\[
i \alpha \left[ \left( U - \frac{\beta}{\alpha} \right) \left( \phi''(y) - \alpha^2 \phi(y) \right) - U'' \phi(y) \right] = \nu \left[ \phi'''(y) - 2 \alpha^2 \phi''(y) + \alpha^4 \phi(y) \right], \tag{F23}
\]

or in dimensionless notation:

\[
(U - c) \left( \frac{d^2 \phi}{dy^2} - \alpha^2 \phi \right) - \frac{d^2 \tilde{U}}{d \tilde{y}^2} \phi = - \frac{i}{\alpha \text{Re}_s} \left( \frac{d^4 \phi}{dy^4} - 2 \alpha^2 \frac{d^2 \phi}{dy^2} + \alpha^4 \phi \right), \tag{F24}
\]
which is called the Orr-Sommerfeld equation with $\tilde{U} = \frac{U}{U_\infty}$ and $\tilde{y} = \frac{y}{\delta_{0.99}}$. The boundary conditions for a boundary layer flow are given by:

- $y = 0 : u' = v' = 0; \varphi = 0, \varphi' = 0$
- $y = \infty : u' = v' = 0; \varphi = 0, \varphi' = 0$

Appendix G: Matlab program to process data

```matlab
clear;
main=1;file='Strip3Pos5';
proberad=0.05;

% reading measured data
pathway= strcat('D:\Stage Belfast\Labview\2905\',file);
[dstore] = dlmread(strcat(pathway,'.dat'),',');
[M,N]=size(dstore);
for loopvar=1:M
    height(loopvar)=dstore(loopvar,1)+proberad;
    freestream(loopvar)=dstore(loopvar,2);
    veldata(loopvar)=dstore(loopvar,6);
    velpuls(loopvar)=dstore(loopvar,7);
end

MaxUe=max(veldata);
Ue99=MaxUe*0.99;
for loopvar=1:M
    if Ue99 > veldata(loopvar);
        delta(main)=height(loopvar);
        endval = loopvar+1;
    else
        break;
    end
end

non_dimU = veldata/Ue99;
non_dimY = height/delta(main);

for loopvar4 = 1:length(non_dimU),
    new_dimU(loopvar4+1) = non_dimU(loopvar4);
    non_dimU1(loopvar4+1) = 1 - new_dimU(loopvar4+1);
    non_dimU2(loopvar4+1) = new_dimU(loopvar4+1)*non_dimU1(loopvar4+1);
    non_dimY2(loopvar4+1) = height(loopvar4)/delta(main);
end

new_dimU(1)=0;  non_dimU1(1)=1;  delst(main) = 0;
non_dimU2(1)=0;  non_dimY2(1)=0;  moment(main) = 0;
```

84
for loopvar4 = 1:length(new_dirnU)-1,
diffheight=non_dirnY2(loopvar4+1)-non_dirnY2(loopvar4);
delst(rnain)=delst(rnain)+(((non_dirnUl(loopvar4+1)+non_dirnUl(loopvar4))/2)*diffheight);
moment(rnain)=moment(rnain)+(((non_dirnU2(loopvar4+1)+non_dirnU2(loopvar4))/2)*diffheight);
end
delst(main)=delst(main)*delta(main);
moment(main)=moment(main)*delta(main);
H(main)=delst(main)/moment(main);
R_theta(main) = (MaxUe*moment(main)/1000)/0.00001455;

lam_Y = 0:0.01:1;
turb_Y = 0:0.01:1;

lam_U = 2*lam_Y - (2*lam_Y.^3) + lam_Y.^4;

for loopvar=1:length(lam_U)
    if 0.99 >= lam_U(loopvar);
        delta_lam(main)=lam_Y(loopvar);
    end
end
turb_U = turb_Y.(1/7);

for loopvar=1:length(turb_U)
    if 0.99 >= turb_U(loopvar);
        delta_turb(main)=turb_Y(loopvar);
    end
end

lam_Y = lam_Y/delta_lam(main);
turb_Y = turb_Y/delta_turb(main);

for loopvar = 1:length(non_dirnY)
    quotient(loopvar)=velpuls(loopvar)/veldata(loopvar);
end

for loopvar = 1:length(non_dirnY)
    if 1 >= non_dirnY(loopvar);
        non_dim_delta(loopvar)=non_dirnY(loopvar);
        delta_quotient(loopvar)=quotient(loopvar);
        endval = loopvar+1;
    else
        break;
    end
end

A=mean(delta_quotient);
```matlab
figure(1);
hold on;
title(['Boundary Layer Profile']);
plot(non_dimU, non_dimY, 'o');
plot(lam_U, lam_Y, 'k--');
plot(turb_U, turb_Y, 'r--');
legend(['Experimental', 'Theory Lam', 'Theory Turb'], strcat('vel =', num2str(MaxUe)), 2);
ylabel('Non-dim height');
xlabel('Non-dim velocity');
axis([0 1.1 0 1.5]);
printlabel = strcat(pathway, 'Fig-1.bmp');
saveas(1, printlabel, 'bmp');

act_vel_lab = strcat('Actual vel =', num2str(MaxUe), ' m/s');
delta_lab = strcat('Delta = ', num2str(delta(main)), ' mm');
deltst_lab = strcat('Delta = ', num2str(delst(main)), ' mm');
mom_lab = strcat('Theta = ', num2str(moment(main)), ' mm');
H_lab = strcat('Shape Factor = ', num2str(H(main)));
R_lab = strcat('R theta = ', num2str(R_theta(main)));

figure(2);
hold on;
title(['Boundary Layer Profile Properties']);
plot(new_dimU, non_dimY2);
plot(non_dimU1, non_dimY2, 'k--');
plot(non_dimU2, non_dimY2, 'r--');
legend('u / Ue', '(1 - u/Ue) X u/Ue', delta_lab, deltst_lab, mom_lab, H_lab, R_lab, act_vel_lab, 0);
ylabel('Non-dim height');
xlabel('Non-dim velocity');
axis([0 1 0 1]);
printlabel = strcat(pathway, 'Fig-2.bmp');
saveas(2, printlabel, 'bmp');

figure(3);
hold on;
title(['Pulsation Profile']);
plot(velpuls, non_dimY, 'o');
ylabel('Non-dim height');
xlabel('Pulsation velocity [m/s]');
axis([0 6 0 1.5]);
printlabel = strcat(pathway, 'Fig-3.bmp');
saveas(3, printlabel, 'bmp');
```

aver=strcat('Average [delta]: u''/u = ', num2str(A));

figure(4);
hold on;
title(['Pulsation velocity/Mean velocity']);
plot(quotient, non_dimY, 'o');
legend('u''/u', aver);

86
ylabel('Non dim height');
xlabel('u'/'u');
axis([0 1 0 1.5]);
printlabel = strcat(pathway,'Fig-4.bmp');
saveas(4,printlabel,'bmp');

Ac(main) = MaxUe;
Ad(main) = delta(main);
Ae(main) = delst(main);
Af(main) = moment(main);
Ag(main) = H(main);
Ah(main) = R_theta(main);

clear H*;clear d*;clear M;clear N;clear U*;clear N*;clear t*;clear l*;
clear n*;clear s*;clear mo*;clear Ma*;clear a*;clear T*;clear veld*;clear f*;
clear vo*;clear h*;clear velocity;clear vell*;

Valfile = strcat(pathway,'Values.txt');
fid4 = fopen(Valfile,'w');
j='Actual_vel';bs='delta';k='0.1 delta';c='delst';d='mom';e='H';
g='R';
fprintf(fid4,'%s %s %s %s %s %s
',j,b,k,c,d,e);
fprintf(fid4,'%5f %5f %5f %5f %5f %5f %5f %5f
',...  
Ac(main),Ad(main),(0.1*Ad(main)),Ae(main),Af(main),Ag(main),Ah(main));
fprintf(fid4,'%s
');
fclose(fid4);
disp(pathway);
clear;

Appendix H: Power spectra
Figure H1: Power spectral density at Strip 1 Pos 5 height 2 mm.
Figure H2: Power spectral density at Strip 1 Pos 7 height 2 mm.
Figure H3: Power spectral density at Strip 1 Pos 9 height 2 mm.
Figure H4: Power spectral density at Strip 2 Pos 5 height 2.5 mm.
Figure H5: Power spectral density at Strip 2 Pos 7 height 2.5 mm.
Figure H6: Power spectral density at Strip 2 Pos 9 height 2.5 mm.
Figure H7: Power spectral density at Strip 3 Pos 9 height 2.5 mm.
Figure H8: Power spectral density at Strip 4 Pos 5 height 18.6 mm.
Figure H9: Power spectral density at Strip 4 Pos 7 height 18.5 mm.
Figure H10: Power spectral density at Strip 4 Pos 9 height 3 mm.
Figure H11: Power spectral density at Strip 5 Pos 5 height 22 mm.
Figure H12: Power spectral density at Strip 5 Pos 7 height 22.1 mm.
Figure H13: Power spectral density at Strip 5 Pos 9 height 3 mm.
References


[53] www.me.umn.edu/courses/me8343/handouts/IntEqns.pdf


