Experimental investigation of re-entry aerodynamic phenomena

Development of non-intrusive flow diagnostics in a Ludwieg tube

F.F.J. Schrijer

Experimental investigation of re-entry aerodynamic phenomena

F.F.J. Schrijer
Experimental investigation of re-entry aerodynamic phenomena
Development of non-intrusive flow diagnostics in a Ludwieg tube

PROEFSCHRIFT

ter verkrijging van de graad van doctor aan de Technische Universiteit Delft, op gezag van de Rector Magnificus Prof. ir. K.C.A.M. Luyben, voorzitter van het College voor Promoties, in het openbaar te verdedigen op woensdag, 27 januari 2010 om 10.00 uur door

Ferdinand Franciscus Johannis SCHRIJER

ingenieur luchtvaart- en ruimtevaart techniek geboren te Geleen.
Dit proefschrift is goedgekeurd door de promotor:

Prof. dr. F. Scarano

Samenstelling promotiecommissie:

Rector Magnificus
Prof. dr. F. Scarano
Prof. dr. ir. P.G. Bakker
Prof. dr. G.M. Carlonmagno
Prof. Dr. Ing. R. Radespiel
Prof. Dr. Ing. M. Raffel
Ir. W.J. Bannink
Ir. J. Muyleart

voorzitter
Technische Universiteit Delft, promotor
Technische Universiteit Delft
Università degli Studi di Napoli Federico II
Technische Universität Braunschweig
Leibniz Universität Hannover & Deutsches Zentrum für Luft- und Raumfahrt
Technische Universiteit Delft
von Karman Institute for Fluid Dynamics

Printed by Proefschriftenmaken.nl, Oisterwijk, The Netherlands
Copyright © 2010 by F.F.J. Schrijver

All rights reserved. No part of the material protected by this copyright notice may be reproduced or utilized in any form or by any means, electronic or mechanical, including photocopying, recording or by any information storage and retrieval system, without the prior permission of the author.

ISBN 978-90-8891-146-0

Cover design by Ferry Schrijver and Bas Heitzer.
Typeset with the \LaTeX Documentation System.
Author email: f.f.j.schrijver@tudelft.nl
Summary

To further improve the design of hypersonic vehicles so that fully reusable systems become a reality, the detailed understanding of the flow field that the vehicle generates is of paramount importance. This thesis deals with the aspects of experimental flow research which includes also the flow facility and the measurement techniques. First a description is given of the investigation and assessment of the Hypersonic Test Facility Delft (HTFD), which is the facility used in the experiments. Thereafter two measurement techniques, particle image velocimetry (PIV) and quantitative infrared thermography (QIRT) are introduced that enable to obtain high quality measurement data of the hypersonic flow field generated by the HTFD. In the remainder of the thesis two important flow phenomena, shock wave boundary layer interaction and boundary layer transition, have been investigated using PIV and QIRT that are of great importance for re-entry vehicles.

The HTFD flow facility is a hypersonic Ludwieg tube capable of generating a range of Mach numbers (typically from 6 to 11) at relatively high unit Reynolds number (from $10^6$ to $10^7 [m^{-1}]$) with a running time in the order of 100 ms. The facility is described theoretically and an experimental evaluation is performed to infer the facility performance for the Mach-7 and Mach-9 nozzles that are also used in the flow investigations described in the thesis. Experiments are performed using conventional techniques such as static and total head pressure measurements and Fay-Riddell heat flux evaluations by means of infrared thermography. Furthermore PIV is used to deduce nozzle boundary-layer parameters as well as the freestream velocity field and the static and total temperature for the Mach-7 nozzle. Finally for the Mach-9 nozzle, stagnation heat flux measurements are performed to obtain the total temperature of the flow. The freestream values were determined experimentally in two different ways and the results showed good agreement. The application of PIV directly allows to obtain the freestream flow field and gives a direct measure for the flow field uniformity (0.2%) and repeatability (0.4%). The static and total temperature calculated from the PIV results showed that there is a large mismatch between the theoretical total temperature (750 $K$) and measured total temperature (579 $K$), which is attributed to heat losses that are present in the throat and nozzle.

The PIV measurement technique is discussed with special emphasis on particle characterisation. The shock test is introduced in combination with an unbiased tech-
nique that allows to infer the particle performance in terms of the relaxation time constant. The procedure is applied to particles having different crystal sizes. It was found that the relaxation time was in the order of $\tau = 2\mu s$ and that it increases with the crystal size. However for the smallest crystal size a longer relaxation time was found which is attributed to excessive agglomeration. Finally a procedure is presented that allows to correct for the particle slip under steady conditions. Using this procedure the particle slip length was reduced with 50%. A non-intrusive heat transfer measurement is introduced that uses thermograms that are obtained using an infrared camera. The physical properties on which the technique is based are discussed as well as the data reduction methods.

Both measurement techniques are used to investigate the flow over a double compression ramp, which is a pertinent problem related to the flow over control surfaces, inlets and ablative surfaces. Two configurations were tested at Mach 7.5, in both cases the first ramp angle was equal to 15° and the second ramp angles were 30° and 45° that generated different Edney type shock interactions namely a type VI and a type IV respectively. Both flow fields were measured using PIV and the typical flow features such as shock angle showed good agreement with theory and schlieren visualizations. Additionally, the measurements were compared to two dimensional CFD computations that showed good agreement. However, the separation region was found to be drastically larger in the computations, especially for the Edney type IV interaction. This was believed to be caused three dimensional flow features, therefore the 15-45° model was investigated using stereo PIV. The experiments showed that the outer flow field is essentially two dimensional. Close to the model surface in the separated region, a large cross flow was measured which causes a decrease of the separated region. Measurements performed using infrared thermography also indicated the presence of streaks that indicate the presence of Görtler vortices.

Roughness induced boundary layer transition was studied in the framework of the EXPERT project. During reentry, due to the large thermal loading local variations in the surface geometry may occur that trigger boundary layer transition. In the experimental investigation the effect of different roughness types on the boundary layer state is observed using infrared thermography at Mach 9.5. The results were then compared to existing empirical transition correlations such as the PANT and shuttle correlation. It was found that two dimensional roughness types such as a step or a wire were ineffective in triggering transition and the classical correlations could not be applied for this case. The three dimensional elements were much more effective in triggering transition and the agreement with the transition correlations was reasonable. In addition a relatively simple transition criterion is proposed based on a critical roughness height Reynolds number. The advantage of this criterion is that it allows to easily scale the flow conditions from the wind tunnel to real flight.
Samenvatting

In de ruimtevaart is de inzet van al dan niet volledig herbruikbare systemen bij hypersonewoertuigen een realiteit geworden. Om dit te bereiken is een theoretisch en experimenteel doordacht ontwerp van primair belang, hetgeen slechts gerealiseerd kan worden indien een gedetailleerde kennis van, en een gedetailleerd inzicht in, het stromingsveld rond het voertuig bestaat. In dit proefschrift wordt ingegaan op het experimentele facet van het stromingsonderzoek. Hiertoe wordt allereerst de faciliteit beschreven waarin de experimenten worden verricht: de Hypersone Test Faciliteit Delft (HTFD). Vervolgens komen twee meettechnieken aan bod die toegepast zijn om een nauwkeurige indruk te krijgen van de stroming in het veld om een model en aan het oppervlak: respectievelijk Particle Image Velocimetry (PIV) en Quantitative Infrared Thermography (QIRT). De bijzonderheid van deze toepassingen ligt in het gebruik ervan enerzijds in het hypersonesnelheidsgebied en anderzijds in een faciliteit met zeer korte looptijd. In het vervolg van het proefschrift worden met behulp van PIV en QIRT twee stromingsproblemen onderzocht die met name voor hypersonewoertuigen van belang zijn: schokgolf grenslaag interactie en grenslaag transitie.

De HTFD is een hypersonewo Ludwieg buis, genoemd naar zijn ontwerper. Met deze faciliteit is het mogelijk om hypersonewo snelheden te genereren voor een aantal discrete getallen van Mach van 6 t/m 11 in combinatie met relatief hoge getallen van Reynolds van $10^6$ tot $10^7$ per meter. De looptijd is voor hypersonetunnel begrippen lang n.l. ca 100 milliseconden. De theoretische werking van de schokbuis met de daarvoor afgeleide formules ter bepaling van de stromingsgrootheden wordt beschreven. Deze grootheden worden geëvalueerd met metingen bij Mach 7 en 9, getallen van Mach die ook in de in dit proefschrift beschreven experimenten aan de orde komen. De metingen van de stromingsgrootheden in de testsectie van de tunnel zijn uitgevoerd met conventionele meetmethoden (druk- en optische metingen) en met gebruikmaking van bovengenoemde geavanceerde technieken PIV en QIRT. PIV is benut in het geval van de nozzle voor Mach 7 bij de bepaling van het snelheidsveld in de ongestoorde stroming en bij de meting van de grenslaagparameters aan de nozzlewand. Het snelheidsveld in de ongestoorde stroming en het getal van Mach verschaf de mogelijkheid om de statische en totale temperatuur af te leiden. Met de nozzle voor Mach 9 zijn warmteoverdrachtsmetingen uitgevoerd eveneens ter bepaling van de totale temperatuur in de stroming. De ongestoorde stromingsgrootheden zoals totale druk, total
temperatuur en getal van Mach zijn op twee afzonderlijke manieren gemeten en de resultaten kwamen goed overeen. PIV laat zien dat voor de ongestoorde stroming de uniformiteit binnen 0,2% blijft en de reproduceerbaarheid binnen 0,4%. De totale temperatuur gemeten met behulp van de PIV resultaten geeft een minder rooskleurig resultaat t.o.v. de theoretische waarde: 579 K t.o.v. 750 K. Dit grote verschil wordt toegeschreven aan het warmteverlies in het constructief zeer complexe deel van de faciliteit tussen opslagbuis en nozzle waar zich de snelopenende klep en de keel bevinden.

De beschrijving van de PIV meettechniek geschiedt aan de hand van een nadere karakterisering van het gedrag van de TiO₂ deeltjes die dienen als seeding. Om het gedrag van de deeltjes met betrekking tot de relaxatielengte te beoordelen is de schoktest gentroduceerd. De toepassing van deze test op deeltjes van verschillende kristalgrootte levert een gemiddelde relaxatietijd van de grootte orde τ = 2 μs. Namaanthe de deeltjes kleine toeneemt, neemt ook de relaxatietijd toe hetgeen verwacht mag worden. Agglomeratie van deeltjes met kleine kristalgrootte veroorzaakt eveneens een toenemende relaxatietijd. Om het verschijnsel dat deeltjes doorslippen (particle slip) in een stationaire stroming te corrigeren wordt een procedure gepresenteerd die de doorsliplengte tot de helft beperkt. De oppervlaktestroming is gemeten met een meetmethode (QIRT) die op niet intrusieve wijze de warmteoverdracht van het model naar de omgeving meet. De methode werkt via thermogrammen verkregen met behulp van een infrarood camera. De fysische eigenschappen waarop de methode berust en de verwerkingstechniek van de meetresultaten zijn in dit proefschrift beschreven.

De meetmethodieken PIV en QIRT zijn toegepast ter bestudering van de hypersonc stroming langs een dubbele compressieknik. Het stromingsprobleem langs dit soort configuraties is van groot belang bij onderdelen van hypersonc voertuigen zoals stuurvlakken, inlaten, ablatieve oppervlaktedelen. Er zijn twee configuraties bestudeerd voor een Mach getal van 7.5: bij beide is de eerste wandhoek identiek (15°), de tweede wandhoek staat respectievelijk onder 30° en 45°. De interacties tussen de schokgolven gegenereerd door de configuratie inclusief de loslatings verschijnselen van de stroming langs de knik veroorzaken verschillende interacties (voor de 15° - 30° en de 15° - 45° resp. de z.g. Edney VI en Edney IV typen).

De PIV resultaten vertonen een goede overeenkomst met kenmerkende grootheden, zoals schokhoeken, sliplijnen, bepaald met de schokgolftheorie, CFD berekeningen en met schlierenvisualisaties. Wel dient te worden opgemerkt dat het loslatgebied in de stromingsovergang bij de knik in de CFD resultaten beduidend groter is dan in het geval van de experimenten, in het bijzonder waar het Edney interactietype IV (15° - 45°) betreft. Dit verschil kan worden geweten aan het feit dat de stroming om het fysische model niet tweedimensionaal is, hetgeen in het CFD geval wel zo is. Om dit te verifiere zijn aanvullende metingen gedaan aan het 15° - 45° model met stereo PIV. Het blijkt dat de buitenstroming in het midden van het model als twee dimensionaal kan worden gezien, maar dat dicht bij het oppervlak in het loslaatgebied een sterke dwarsstroming bestaat die zorgt voor een zekere verlichting van dat gebied waardoor het in omvang reduceert. De stroming over de compressieknik modellen is ook bestudeerd met gebruikmaking van de infrarood thermografie. De resultaten hiervan geven additionele informatie over de oppervlaktestroming; er blijken in stromingsrichting
regelmatig verdeelde strepen (Engels: streaks) zichtbaar in de thermogrammen, hetgeen een indicatie geeft van het bestaan van Görtler wervels.

De infrarood thermografie is ook toegepast voor het tweede stromingsprobleem van dit proefschrift: de bestudering van grenslaagtransitie veroorzaakt door oppervlakteruwwheid. Gedurende de terugkeer van een ruimtevoertuig naar de aarde kunnen door de grote warmtebelasting lokale veranderingen in de geometrie optreden die een omslag van de grenslaag van laminair naar turbulent teweeg brengen. Om na te gaan in hoeverre dit nadelige gevolgen heeft voor de re-entry vlucht heeft in het kader van het EXPERT project een experimentprogramma plaats gevonden om met infrarood thermografie te onderzoeken hoe verschillende vormen van oppervlakteruwwheid (geïsoleerd, twee- en drie dimensionaal) de toestand van de grenslaag beïnvloeden bij Mach 9.5. De resultaten zijn vergeleken met bestaande klassieke empirische transitie correlaties voor re-entry toepassingen: shuttle correlatie, PANT correlatie. Uit de meetresultaten valt op dat twee dimensionale ruwheiden, bijvoorbeeld een draad of een stap in het oppervlak, weinig effectief zijn in het veroorzaken van transitie. De klassieke correlatiemethoden bieden dan ook geen soelaas in de voorspelling van transitie. Drie dimensionale ruwheids elementen in het oppervlak zijn veel effectiever en de meetresultaten passen goed in de correlatiediagrammen. Een relatief eenvoudig transitiecriterium wordt geïntroduceerd gebaseerd op een kritiek getal van Reynolds met als lengteschaal de kritieke hoogte van de ruwheid waarvoor transitie wordt vastgesteld. Dit criterium maakt het mogelijk windtunnel condities eenvoudig op te schalen naar vrije vlucht situaties.
Contents

Summary iii

Samenvatting v

1 Introduction and relevance 1
  1.1 Historical perspective ........................................... 1
  1.2 Hypersonic vehicle aerodynamics ............................. 5
    1.2.1 The hypersonic environment .............................. 5
    1.2.2 Aerodynamic testing ........................................ 6
    1.2.3 Critical aerothermodynamic phenomena ................... 7
    1.2.4 Programs, vehicles and missions ......................... 8
  1.3 Ground based testing .......................................... 10
    1.3.1 Flow facilities ............................................. 11
    1.3.2 Measurement techniques ................................. 12
  1.4 Objective and scope ........................................... 13

2 Experimental facility 15
  2.1 Introduction .................................................. 15
  2.2 Ludwig tube flow facility .................................... 17
    2.2.1 Operating principle ....................................... 17
    2.2.2 Nozzle flow ............................................... 21
    2.2.3 Free stream conditions .................................... 25
    2.2.4 Stagnation heat flux measurements ....................... 29
    2.2.5 Operational envelope ..................................... 30
    2.2.6 Condensation effects ..................................... 31
  2.3 Implementation of measurement techniques ................... 32
    2.3.1 Particle Image Velocimetry (PIV) ......................... 33
    2.3.2 Quantitative Infrared Thermography (QIRT) ............. 34
    2.3.3 Schlieren .................................................. 35
    2.3.4 Synchronization ........................................... 35
### 3 Particle Image Velocimetry

- **3.1 Introduction** .................................................. 37
- **3.2 Measurement principle and data processing** .............. 37
  - 3.2.1 Flow seeding .................................................. 38
  - 3.2.2 Particle imaging and illumination .......................... 39
  - 3.2.3 Data processing ............................................... 41
  - 3.2.4 Stereoscopic PIV ............................................. 43
- **3.3 Particle characterisation** .................................... 44
  - 3.3.1 Particle response ............................................ 45
  - 3.3.2 Determination of particle time response by means of a shock wave test ............................................. 46
  - 3.3.3 Assessment of the particle time response .................. 47
- **3.4 Particle slip compensation** .................................. 49
  - 3.4.1 Application to double ramp flow ......................... 50

### 4 Quantitative Infrared Thermography

- **4.1 Introduction** .................................................. 53
- **4.2 Infrared theory** ............................................... 53
  - 4.2.1 Thermal radiation ............................................ 53
  - 4.2.2 Real emitters ................................................ 54
  - 4.2.3 Environmental disturbances ................................. 56
- **4.3 Infrared measurement systems** .............................. 57
  - 4.3.1 Agema 880 LWB system ..................................... 58
  - 4.3.2 CEDIP Titanium 530L system ............................... 59
  - 4.3.3 Calibration procedure ..................................... 59
  - 4.3.4 Environmental disturbance correction .................... 61
- **4.4 Heat flux data reduction** .................................... 61
  - 4.4.1 Heat flux sensors .......................................... 61
  - 4.4.2 One dimensional semi-infinite model ..................... 62
  - 4.4.3 Data reduction techniques ................................ 64
  - 4.4.4 Error sources .............................................. 65
  - 4.4.5 Heat flux normalization .................................... 66

### 5 Double compression ramp flows

- **5.1 Introduction** .................................................. 67
- **5.2 Wind tunnel model** .......................................... 70
- **5.3 15-30 degree ramp flow** .................................... 70
  - 5.3.1 Comparison with CFD computations ....................... 73
- **5.4 15-45 degree ramp flow** .................................... 75
  - 5.4.1 Comparison with CFD computations ....................... 77
  - 5.4.2 Unsteady flow features and turbulence .................. 78
- **5.5 Three dimensional investigation of the flow over the 15-45 ramp model**  
  - 5.5.1 Experimental setup ....................................... 80
  - 5.5.2 Outer flow region .......................................... 81
  - 5.5.3 Inner flow region, first ramp ........................... 84
  - 5.5.4 Inner flow region, second ramp ......................... 87
  - 5.5.5 Flow field overview ....................................... 89
6 Roughness induced boundary layer transition 93
   6.1 Introduction .................................................. 93
   6.1.1 Shuttle correlation ....................................... 94
   6.1.2 PANT correlation ......................................... 95
   6.1.3 $Re_k$ correlation ......................................... 95
   6.2 Experimental setup and data reduction .................... 96
   6.3 IR measurements ............................................. 97
   6.4 Results and extrapolation to flight ......................... 99
   6.4.1 PANT and Shuttle correlation ............................ 99
   6.4.2 $Re_k$ correlation and extrapolation to flight .......... 101

7 Conclusions 103
   A HTFD running time derivation 107
   B Derivation of $\frac{d_{\text{hot}}}{d_{\text{cold}}} = \left( \frac{T_{\text{cold}}}{T_{\text{hot}}} \right)^{\frac{1}{2}}$ 111

References 115

List of publications 123

Acknowledgements 127

Curriculum Vitae 129
Chapter 1

Introduction and relevance

1.1 Historical perspective

Figure 1.1: Salvaged Apollo capsule after re-entry

Figure 1.2: Soyuz TM capsule

During the second world war, the development of high-speed vehicles began by the advent of the first rocketeers, which generated increased interest in high speed aerodynamics. The first rocket capable of reaching supersonic speeds was the A4, later renamed to V2 for military purposes. During this time supersonic and hypersonic flow research intensified and consequently the first supersonic/hypersonic wind tunnel was realized in Peenemiinde, Germany that was able to generate flows more than four times the sound speed. After the war the German scientists moved to Soviet Union and United States where they continued to work on rocketry and reentry technology driven by its relevance for military applications. On 4 October 1957 the first man-made satellite Sputnik 1 was launched into orbit. After 3 months the batteries were depleted and the Sputnik 1 could not hold its orbit and so constituted the
first orbital reentry in history, although it completely burned. The USA, startled by the achievement of the USSR, accelerated its space research and the space race officially started. Soon after Sputnik 1, Sputnik 2 was launched carrying a dog into space. However the capsule was not intended to make a safe reentry. In 1958 the USA launched a squirrel monkey into space which subsequently also made the first successful reentry, unfortunately the parachute failed and the capsule was lost. Then, in April 1961 Yuri Gagarin became the first human in space when he entered orbit in the Vostok 1, he safely returned to earth 108 minutes later. After the successes of the USSR, the USA started the Apollo human space-flight program with the goal of putting man on the moon, this achievement was finally accomplished on July 20, 1969.

In the Apollo program and its predecessors Gemini and Mercury, the reentry vehicle, which was part of the transportation system was capsule shaped because of its large volumetric efficiency. The capsule geometry has a low aerodynamic lift-to-drag ratio which leads to a basically ballistic reentry trajectory resulting in a deeper penetration of the capsule into the atmosphere before deceleration would occur. This results in high heat loads since the air density is higher at low altitudes. Fortunately because of the limited time duration of the high heat load opens the possibility to use heat sink approaches in combination with surface ablation for the thermal protection system. In case of the Mercury and Apollo vehicles, a small L/D (approximately 0.15 for Mercury and 0.35 for Apollo) was implemented by offsetting the center of gravity in order to have some maneuverability (cross range) and to alleviate the maximum g-loads and peak heating rates. In the Sovjet Union, the reentry vehicles used to ferry kosmonauts to and back from orbit also featured capsule geometries such as the Vostok which carried Gargarin, followed by Voskhod and Soyuz of which the latter, in its fourth generation, is still in service today to transport astronauts from the International Space Station back to earth.

![Figure 1.3: X-15 just after launch](image)

In parallel to the design and development of capsule shaped reentry vehicles, programs were established to investigate the possibilities of lifting reentry vehicles for high L/D ratio. The advantages of lifting reentry vehicles is that the cross range is larger which improves mission flexibility and reduction of maximum g-loads. Additionally, the maximum heating rates are reduced since the vehicle starts decelerating
in the low density part of the upper atmosphere. This allows for the application of simpler or even reusable thermal protection systems. However, the vehicle complexity increases with increasing L/D and although the overall heat loads are reduced, due to sharper edges the local heat loads (and operating temperatures) may increase critically. Moreover, the total heating time increases which reduces the heat sink benefits. Despite the disadvantages, lifting reentry bodies were (and still are today) considered to be the future of space transportation systems.

The first winged vehicle to travel at high speed (Mach 6.7) was the North American X-15 (figure 1.3) which provided a wealth of knowledge on aerothermodynamics. At the same time the X-20 Dyna-Soar (see figure 1.4) was conceived as a space plane that could be used in a variety of military missions. During the X-20 project the ASSET (Aerothermodynamic Elastic Structural Systems Environmental Tests) vehicle was used to test the superalloy heat shield for the X-20.

![Figure 1.4: X-20 Dyna-Soar artist impression](image1)
![Figure 1.5: MiG 105 'Spiral'](image2)

At the same time in the USSR the Spiral project (see figure 1.5) was started in order to develop their version of a space plane which showed quite some analogies to the Dyna-Soar project. Finally the Spiral programme was stopped and the knowledge was used in the development of the BOR series which, in turn provided data for the Buran project (the counterpart of the Shuttle Transportation System).

When the X-20 project was cancelled the ASSET vehicles were used to build a data base that could be used for the development of future space vehicles, such as the Shuttle Transportation System. Subsequently the PRIME (X-23) project was started, which was intended to study the effects of maneuvering reentry. Hereafter the PILOT program was started by USAF-NASA in order to investigate the supersonic, transonic and subsonic behavior of lifting bodies. The X-24A developed in the program, based on the X-23, was used as the later baseline for the X-38 Crew Return Vehicle. In 1972 it was decided to proceed with a reusable Space Shuttle system. In the programs conceived in the 1960’s all transportation systems were planned to be fully reusable, however, it appeared that this was not possible by the current state of technology and available funding. Therefore the final Space Transportation System featured a reusable Space Shuttle Orbiter (figure 1.6) with solid rocket boosters that are recovered after usage. The main external tank is expendable since it breaks up before impact in the Indian ocean. The Space transportation system has now been in operational use since April 1981, and has delivered many payloads to low earth orbit.
(e.g., Hubble Space telescope, parts of the International Space Station, SpaceLab). To date this has been the only winged reentry vehicle that has been used on a regular basis. In practise it was found that the Space Transportation System could not satisfy the cost and utility goals set for the program.

For the successor of the Space Transportation System a fully reusable single stage to orbit (SSTO) system was envisioned which is a revolutionary concept within the limits stated by Tsiolkovsky. This resulted in the start of the X-33 project as a sub-scale demonstrator for the VentureStar and the X-30 National Aero-Space Plane (NASP). Both projects were cancelled due to technical problems and budget concern. A scaled-down unmanned version of the X-30, the X-43 Hyper-X is currently being used to develop and test scramjet engines.

Since the Space Transportation System is being retired in 2010 a successor is needed in order to retain manned access to space. In 2004 The Vision for Space Exploration was announced by the president of the United States, George W. Bush. These plans resulted in Project Constellation, a program to create a new generation of spacecraft for human spaceflight. The reentry module called Orion (figure 1.9) will
feature a crew module similar in shape but slightly larger than the Apollo command module and it will be reused partially for approximately 10 times. In Russia and Europe the same trend is observable where a collaboration is foreseen on the crew space transportation system (CSTS) which will also feature a capsule-like design reminiscent from the Soyuz, which this has been favoured above the Kliper in Russia and Hermes in Europe, the latter two were similar to the X-38 crew return vehicle. Furthermore an adaptation of the ATV (Automated Transfer Vehicle) that successfully docked with the ISS in March 2008 is being developed. This vehicle, called CARV also features a lifting capsule shape. Also SpaceX, a company that won a NASA COTS (Commercial Orbital Transportation Services) contract has a capsule shaped ISS crew and resupply vehicle (Dragon, figure 1.10) under development.

Although capsule shaped vehicles prove to be more economic and realizable in the short term, winged re-entry vehicles are still considered to be the future in space transportation since they provide the performance gains that someday truly make cheap access to space possible.

1.2 Hypersonic vehicle aerodynamics

During its descent into the earth atmosphere a reentry vehicle encounters varying flow situations from highly hypersonic to subsonic. The current section globally discusses the hypersonic flow environment that will be encountered, how it can be investigated and what the critical flow phenomena are.

1.2.1 The hypersonic environment

An exact definition of hypersonic flow is difficult to establish, generally it is given to a flow that exceeds Mach 4 or 5. It covers a flow regime where certain physical phenomena become progressively more important with increasing Mach number. At these Mach numbers the shock angles become small and therefore shocks move closer toward the surface. Additionally the relative boundary layer thickness increases due to the lower overall density. Combining these two effects results in a close interaction of the two phenomena generally referred to as viscous interaction where the shock wave and boundary layer are merged. Furthermore due to the higher Mach numbers
the subsonic portion of the boundary layer becomes smaller which has a significant impact on boundary layer characteristics. In these conditions the compressibility effects cause an enthalpy or thermal boundary layer to be present which causes that the heat transfer within the boundary layer can no longer be neglected. The transition from supersonic to hypersonic flow is strictly speaking not only determined by the Mach number but also by high temperature and rarefaction effects that start to have an influence on the flow field. Among the most important effects are the changing thermodynamic properties (ratio of specific heats, $\gamma$) and non-ideal gas effects (dissociation, ionization) as well as the influence of the wall surface material (catalicity). This is the regime where aerodynamics and thermodynamics also meet chemistry. Due to the inclusion of these effects the flow even in case of a subsonic Mach number can deviate to a large extent from "classical" gasdynamics. For example it is known that in the region behind a bow shock, where high temperatures occur, the effective $\gamma$ is reduced due to dissociation and ionization. This leads to a smaller shock standoff distance which is governed by the subsonic flow behind the strong portion of the shock. In turn the shock standoff distance has a large effect on the aerodynamic coefficients of the geometry under consideration. Another effect is dissociation which absorbs energy from the flow and through which the local temperature effectively decreases, however this energy is released as heat again downstream which results in a local increase in temperature. At the wall the surface material may act as a catalyst (depending on the material) causing a recombination of the dissociated species resulting in an increase in the surface heat transfer.

In figure 1.11 an altitude-velocity diagram is given for different classes of reentry vehicles. In the figure the regions are indicated where specific phenomena occur. At an altitude above approximately 100 km the flow field cannot be regarded as continuous anymore. When the mean free path length between the air molecules becomes of the same order as the vehicle characteristic dimensions the flow is considered to be rarified. For a vehicle travelling at velocities in excess of approximately 4 km/s, the total temperature downstream of the bow shock becomes so high that real-gas effects are expected to occur. Finally, when a vehicle descends into the atmosphere the density increases and, since the vehicle is decelerating, the velocity decreases. However since the increase in density is several orders of magnitude larger than the decrease in velocity, the Reynolds number will effectively increase. Due to the increase in Reynolds number the boundary layer, which is laminar at high altitudes will become turbulent at lower altitudes. Generally this transition will occur at approximately 50 km and Mach 10 depending on the re-entry trajectory.

### 1.2.2 Aerodynamic testing

In general hypersonic flow investigation knows three different approaches. The first is experimental testing in ground based facilities. As is generally known, it is difficult and even nearly impossible to duplicate the hypersonic flow and boundary conditions as they are encountered in re-entry flows or in sustained hypersonic flight. Therefore similarity parameters specific to the flow field under study are duplicated as they are encountered in real flight. Two important similarity parameters are the Reynolds $Re$ and Mach $M$ number that are related to the flow field. Additionally there are parameters that depend on the fluid physical properties such as the specific heat ratio
\[ \gamma \text{ and the Prandtl Pr number. In special cases, when the flow density becomes low, it cannot be regarded as being a continuum anymore and an additional similarity parameter must be introduced. The amount of rarefaction is expressed by the Knudsen number which relates a relevant length scale } l \text{ to the mean free path between the air molecules } \lambda, \quad Kn = \frac{\lambda}{l}. \] For high temperatures, the fluid cannot be regarded as a perfect gas anymore since chemical reactions (e.g. dissociation and ionization) start affecting the flow, this is normally indicated by the stagnation enthalpy } H_0. 
\[ \text{For air, when the stagnation enthalpy exceeds approximately } H_0 > 1MJ/kg, \text{ it starts becoming important. Finally also the wall-to-free stream or boundary layer edge temperature } T_w/T_c \text{ is important since it affects the density in the boundary layer and therefore the local boundary layer characteristics.} \]

A second approach is the simulation of the flow field by means of computational fluid dynamics. Although the capability of hypersonic flow simulations increases steadily, applicability and accuracy is still restricted to relatively simple geometries and flow situations. This is not only because of the limited capability (memory/computational power) of computers but also due to a lack of understanding on flow modeling. Especially in cases where flow transition (laminar to turbulent) and chemical effects play an important role.

Therefore, as a third branch of hypersonic flow investigation, experimental test vehicles are used, which are capable of duplicating almost all important flow parameters. However, this type of testing is expensive and its flexibility is limited.

### 1.2.3 Critical aerothermodynamic phenomena

The most important design parameters in the development of a hypersonic vehicle are the aerodynamic forces and moments as well as the local heating rates. A vehicle must have a predetermined behaviour of its drag coefficient in order to optimize the

---

Figure 1.11: Typical flight regime and most important aerothermodynamic phenomena [36]
trajectory for heating rates or g-forces, together with a minimum lift coefficient that is required to achieve a desired cross-range. In combination with the local heating rate, the surface temperature is paramount to ensure the structural integrity of the vehicle.

The state of the boundary layer that is developing over the surface of a vehicle determines the amount of heat transfer and is therefore vital. For example a turbulent boundary layer features a surface heat transfer that is three times as high as that of a comparable laminar boundary layer [4]. Furthermore, boundary layer transition may affect vehicle attitude and stability due to for example asymmetric transition which may cause a yawing moment or a change in the effectiveness of control surfaces. In the design of hypersonic vehicles, transition prediction is solely based on the usage of empirical correlations that are calibrated using wind tunnel measurements since CFD methods are not yet capable to make accurate predictions. For this reason it is important to develop good measurement techniques that can be used to measure the onset of transition. At the same time efforts must be taken to both increase the knowledge on the applicability of the current correlations and develop new ones that have a wider application area.

For winged vehicles and supersonic inlets a pertinent phenomena is the occurrence of shock wave boundary layer interaction, which may lead to considerable unsteady pressure loads and high localized heating. The accompanying features have been widely investigated and reported in literature [26], albeit that they are still not completely understood and their prediction by numerical models is still limited, in particular regarding the time dependent character of pressure and heat loads. The classical way of investigating these interactions is by means of surface measurement techniques (oil flow, pressure taps and heat flux sensors), which enable to get a general understanding of the flow field. However, in order to further improve the understanding, quantitative flow field measurements are needed that can relate the off-surface phenomena to the resulting heat fluxes and pressure loads.

The above mentioned phenomena will be directly addressed in the current investigation. In addition also chemical reactions and related phenomena such catalicity effects and shock layer radiation have a significant effect on flow field characteristics and are critical factors in the design of hypersonic vehicles.

1.2.4 Programs, vehicles and missions

The vehicles and missions started over the recent decades may be divided into three different categories based on the maturity and goal of the project. First there are the programs on experimental test vehicles; these programs have the objective to investigate basic aerodynamic, structural and production principles which are used in re-entry technology. Some european examples of these programmes are: DART, EXPERT and ARD. The second type of programs consist of technology demonstrators conceived to mature technology and to yield experience and insight in the integration of the different subsystems (IXV, X33, FIRE II). The third category is the utilization vehicle, which uses technologies that are mature and proven. These are the vehicles that are conceived using knowledge of experimental vehicles, technology demonstrators, wind tunnel testing and CFD. Examples of these vehicles are the Space Shuttle Orbiter, the Apollo capsule and the Russian Soyuz.
In Bertin and Cummings [15] a diagram is introduced that inhibits the rationale behind experimental test vehicles and technology demonstrators in the development of utilization vehicles. The diagram (figure 1.13) shows the different states of knowledge that are applicable in the development of hypersonic vehicles and programs. The upper right hand corner describes the knowledge that is obtained from theoretical considerations, wind tunnel testing and previous experiences. In the design of a new vehicle, it is the goal to advance towards the left upper side where new capabilities that we are aware of have to be achieved by new knowledge and targeted research, which is done by means of wind tunnel testing, CFD and experimental vehicles. However, each new but also old technology brings with it unknowns of which there is no awareness (bottom left). An example of this is the catastrophe with the Space Shuttle Orbiter Colombia where the impact of a piece of insulating foam detached from the fuel tank during ascent damaged the heat shield on the wing leading edge, which turned out to be fatal for the vehicle during reentry. In order to prevent this pitfall technology demonstrators are used.

The above described development rationale is further illustrated by the development process of the Space Shuttle. Figure 1.14 shows the calendar containing the projects that contributed in the development of the Space Shuttle. The projects X-15, X-20 Dyna-Soar and ASSET were the experimental test vehicles that were used to increase the overall knowledge level on re-entry technology. The technology demonstrator programs X-23 Prime and X-24 focused on integrating and demonstrating re-entry technologies.
It is well known that simulation of the aerothermodynamic environment of reentry is only partly possible in ground based facilities. However, the specific advantage and motivation for using such facilities nonetheless, is that investigations can be performed under controlled and repeatable conditions. Nowadays ground based test facilities are used for three main purposes: first to provide development of physical understanding and modeling of hypersonic flows, secondly to provide specific vehicle performance data such as lift, drag and structural heating and third to produce data for CFD validation. Since complete simulation of the flow field is not possible the flow phenomena under study determine the type of flow facility. For example when investigating the Space Shuttle Orbiter at high angle of attack the flow phenomena are dominated by the blunt nosed entropy layer, whereas the Mach number is not a very significant parameter. However when going to smaller angles of attack the Mach number does become an important parameter [14].
1.3.1 Flow facilities

Different types of wind tunnels are used in the investigation of high speed flows depending on the desired Mach number, Reynolds number and total enthalpy. In case of supersonic flow fields \(1.2 < M < 4\) mostly blowdown, indraft or continuous wind tunnels are used, in this range temperature effects on the fluid properties (real gas effects) are limited and a high total enthalpy is not needed. For the investigation of hypersonic flow fields \(M > 4\), temperature effects may become important requiring a proper modelling of the stagnation enthalpy, requiring high total enthalpy facilities. This is achieved by energizing the flow through an increased total temperature, which can be achieved by different means. In blow down wind tunnels an (electric) heating system can be used to achieve a storage temperature typically up to 1000 K. Since non-perfect gas effects only start to become significant above 1000 K, these facilities are generally called cold hypersonic facilities and the total temperature is predominantly applied only to avoid undesirable effects in the test section such as flow condensation. In order to increase the total temperatures further other types of tunnels were developed. The mostly applied tunnels are shock tunnels where for example a free running piston and a diaphragm is used to create a shock in a stagnant gas. Due to the passing shock this gas is superheated and acts as the reservoir condition for the nozzle and test section located downstream. Generally the test times of shock tubes are very short, they are in the order of a few milliseconds. Another type of facility capable of generating a high enthalpy flow is the HotShot wind tunnel where an electrical discharge (arc) in the settling chamber is used to create high temperature reservoir conditions. Although the free stream temperature is high in this type of facility, the free stream pressure is rather low. For a complete review of hypersonic facilities the reader is referred to Lukasiewicz [53].

As has already been mentioned, the flow phenomenon under investigation dictates the choice of the flow facility. When the Reynolds number has a large effect on the flow, normally cold hypersonic facilities are used. In case where the determination of forces
and moments are required it is generally important to correctly simulate the Mach number. As the flow field temperature increases (and thus generally also the Mach number) the correct temperatures (effect of changing $\gamma$) should be simulated rather than the Mach number. Furthermore the principle of Mach number independence can be used, which states that at high Mach numbers, certain aerodynamic quantities such as pressure coefficient, lift and drag coefficient, shock shape and Mach wave patterns become independent of Mach number [5], so that Mach number duplication is no longer essential. The experimental facility used in the present study is a Ludwieg tube, which can be considered a cold hypersonic facility. A detailed explanation of the working principle is given in chapter 2.

1.3.2 Measurement techniques

Associated with the capability to duplicate the required flow parameters in ground based facilities, a successful flow investigation largely depends on the measurement capability. A multitude of measurement techniques is available where we may distinguish between qualitative and quantitative, intrusive and non-intrusive and point or field measurements.

Standard non-intrusive measurement techniques in compressible flow facilities are schlieren and shadowgraph visualization methods [80]. The setup is relatively simple and provides a good visualization of the overall flow features. The measurement principle is based on light propagation through refractive media (density field in case of a compressible flow); it integrates along the line of sight, therefore, it is unsuitable for three dimensional flows or complex geometries. Furthermore it only provides qualitative data such as the approximate position of a shock wave, expansion fan or shear layer. Another qualitative measurement technique is flow visualization by means of a contrast fluid, which can be used to monitor mixing phenomena. An example is water injection into the free stream that condenses in low temperature regions, however, care must be taken not to influence the flow behaviour. Finally in high speed (high enthalpy) flow the natural luminosity of the flow can be used in order to visualize features such as shocks and boundary layers [83].

A non-intrusive technique that provides quantitative data is interferometry, it uses phase information in a coherent light beam to measure the flow density. However as with the schlieren technique the information is integrated along the line of sight. In case of relatively low density flows laser induced fluorescence (LIF) may be applied in order to obtain temperature, species concentration or flow velocity data [83]. For higher flow densities CARS (coherent anti-Stokes Raman spectroscopy) may be appropriate for species concentration and temperature measurements [83]. In low density gases the electron beam fluorescence technique can be used for density measurement and vibrational and rotational temperature measurement. Furthermore velocity data can be obtained by means of Laser Doppler Velocimetry (LDV). A major drawback of this measurement technique is that it has a low spatial data yield and therefore is of limited use in short duration facilities. The problem of low spatial data yield may be solved by using a planar measurement techniques such as Particle Image Velocimetry (PIV), Doppler Global Velocimetry (DGV) or Doppler Picture Velocimetry (DPV) [79].
In the present study the development and application of particle image velocimetry is described which enables to obtain the spatial distribution of the velocity field. During the last two decades particle image velocimetry has become a standard measurement technique in the subsonic flow regime and more recently has also been applied to supersonic and hypersonic flow fields. The issues related to the application of PIV in high speed flows such as recording speed and tracer particle accuracy were addressed by Moraitis and Reithmüller [58] and Kompenhans and Höcker [47]. With the advent of fast interframe CCD cameras the digital version of the technique was successfully applied in the supersonic and hypersonic flow regime [35], [71], [76].

Furthermore, another category of measurement techniques exists that rely on the presence of probes in the flow field and are therefore considered intrusive. The most important ones are the five-hole probe that is used for velocity measurements, pitot tubes to measure the Mach number and hot-wires and hot-films to measure velocity and temperature.

The above described measurement techniques provide flow field data, however in flow investigations and design of aerodynamic vehicles the surface properties are as equally important. A qualitative view of the surface flow can be obtained using oil flow investigation, the presence of separation lines, reattachment points and transition locations is visualized by means of removal and accumulation of oil. Quantitative surface measurement techniques such as pressure, thermocouple and heat flux measurements are in the classical implementation discrete, point wise measurements. More recently pressure sensitive paints (PSP) have been developed that allow for a more continuous spatial measurement of the surface pressures however at the expense of a reduced measurement accuracy. For surface temperature measurements also temperature sensitive paints, liquid crystals and infrared thermography may be applied. Among these thermometry techniques, infrared thermography is the most suitable since it is most sensitive [48] and generally does not require a special coating. However optical access in the infrared wavelength regime must be provided.

1.4 Objective and scope

Since shock wave boundary layer interaction and roughness induced boundary layer transition are important phenomena in the design of (winged) reentry vehicles, see section 1.2.3, the current investigation addresses these two aspects, focussing on the investigation of ramp induced shock wave boundary layer interaction and surface roughness induced boundary layer transition. These investigations are facilitated by a careful calibration of the flow facility and the application of state-of-the-art measurement techniques.

Chapter 2 of the thesis discusses the experimental facility used in the investigations. It starts with a description of the facility followed by an experimental investigation of the basic flow process; properties such as pressure and temperature losses and nozzle boundary layer thickness are investigated. Subsequently an assessment of the free stream conditions and repeatability is given. The last part of the chapter deals with the implementation of the different measurement techniques into the flow facility.

In chapter 3 the particle image velocimetry (PIV) technique is treated. The chapter
concentrates on the measurement principle and the data reduction technique. Followed by a description of the phenomenon and physics of particle slip, it discusses how particle performance can be assessed and finally a strategy is proposed that may account for the particle slip phenomena under steady flow conditions.

Chapter 4 deals with the qualitative infrared thermography measurement (QIRT) technique. A short introduction is given on radiation theory which is essential for the measurement principle. Hereafter the infrared data acquisition systems are treated that are used to measure the surface temperature. Finally the data reduction techniques are discussed to obtain the surface heat flux from the measured surface temperature values.

The measurement techniques described in chapters 3 and 4 are applied to a double compression ramp in chapter 5. Two configurations are studied, a 15-30 degree and a 15-45 degree double ramp, with the emphasis on the flow field in the center plane of the model. Thereafter the three-dimensional flow field over the 15-45 degree model is investigated with emphasis on the flow field close to the model surface.

The investigation of roughness induced boundary layer transition is treated in chapter 6 where the influence of surface roughness elements is studied in the framework of the EXPERT project. Quantitative infrared thermography is used to assess the maximum allowable surface step before boundary layer transition is triggered. In addition to a two-dimensional step-like disturbance also two-dimensional wire disturbances and three-dimensional disturbances are investigated.

Finally chapter 7 closes the thesis describing the conclusions following from the current study.
Chapter 2

Experimental facility

2.1 Introduction

The Hypersonic Test Facility Delft uses the Ludwig tube wind tunnel concept, which was first conceived by H. Ludwig [52] in 1955 and was originally intended as a low cost alternative for subsonic/transonic testing at high Reynolds numbers. The advantage of a Ludwig tube facility with respect to other types is the ability to generate a low turbulence uniform flow by placing a fast acting valve downstream of the test section. Later its use was extended for hypersonic applications [39], [40]. In this respect the appeal of this kind of facility for hypersonic testing is the long run time (0.1 to 0.2 seconds) compared to other short duration facilities (e.g. shock tubes, piston tubes) and high unit Reynolds numbers (order of $5 - 50 \times 10^6 \, m^{-1}$). The latter is a consequence of the relative low free stream total temperature. Wind tunnels operating above a free stream Mach number of 4 need to heat the flow in order to prevent condensation effects in the free stream. For classic blow down wind tunnels this means that large heaters are required with an accompanying high energy demand to in-line heat the flow. Since in the Ludwig tube the flow is heated off-line the energy requirements are less demanding and more simple heating systems may be used.

The Ludwig tube consists of four principal elements (see figures 2.1 and 2.2):

- Storage tube, where the fluid is pressurized at temperature controlled conditions
- Conical nozzle, which is separated from the storage tube by a fast acting valve. The convergent-divergent nozzle accelerates the gas from quiescent to hypersonic conditions depending on the area ratio between the throat and test section.
- Test section, where the flow reaches the desired conditions and the model is located.
- Vacuum discharge tank, that maintains a large pressure ratio across the nozzle in order to start and sustain the flow conditions.

When the valve opens, air flows from the storage tube through the nozzle into the test section. The expansion ratio of nozzle to test section determines the Mach number.
in the test section.
The HTFD was conceived to be a flexible hypersonic wind tunnel capable of producing relatively high Reynolds numbers and a range of Mach numbers. In additionally the design of the facility is such that the operating costs are minimized (energy and air usage). The basic wind tunnel design was made by Hyperschall Technologie Göttingen [34] (HTG) in Germany. Sister facilities are in use at the HTG institute itself and at the German university of Bremen. Other hypersonic Ludwig tubes are the facility at Braunschweig university [95] in Germany and the Boeing/AFOSR Mach-6 quiet tunnel at Purdue University [45] in the U.S. The layout of the Bremen Ludwig tube is similar to the HTFD while the Braunschweig Ludwig tube uses a single straight tube and has a larger test section. The Purdue University facility has a slightly different setup since it is specially developed for boundary-layer transition research. The Delft, Braunschweig and Bremen wind tunnels have a fast acting valve located upstream of the nozzle while in the Purdue facility a diaphragm is located downstream of the test section. In the latter facility boundary-layer suction is applied at the throat to achieve a laminar boundary layer along the nozzle wall which reduces the noise in the freestream. However, this is at the cost of larger air usage (the whole test section must be pressurized before the run) and the need of a larger vacuum tank. Furthermore, thicker windows are needed to withstand the high pressures, which makes optical access more difficult.

![Image of the hypersonic wind tunnel](image)

**Figure 2.1: Hypersonic Test Facility Delft**

In the current chapter the operation principle of the facility is discussed, along with the characterization of the flow conditions and implementation of the measurement techniques.
2.2 Ludwieg tube flow facility

In this section a description is given of the operating principle of the Ludwieg tube. Theoretical estimates are given for the running time, free stream total temperature and free stream total pressure for given basic dimensions and gas storage conditions. Furthermore measurements are performed to assess the performance of the facility and to characterize the flow conditions in the test section. An accurate determination of the free stream characteristics (Mach number, total temperature, total pressure) is made for a specific flow operating condition and the Mach-Reynolds envelope is presented as obtained from the facility performance envelope. Additionally the limit for free stream condensation is addressed which is especially pertinent for cold hypersonic facilities.

2.2.1 Operating principle

The air used in the experiments is stored in the storage tube under high pressure and high temperature. A fast-opening valve separates the storage tube from the throat section and subsequent nozzle and test section. When the valve is opened the stored air flows through the nozzle. As a result an expansion wave travels upstream into the storage tube. The flow conditions behind the expansion wave act as reference stagnation conditions for the flow in the test section. The valve opening is relatively quick [95] and the process may be regarded as impulsive, therefore the expansion wave may described by a centered expansion wave. The process is shown in figure 2.2 where a sketch is given of facility together with the wave pattern in a t, x-diagram. The first and last characteristic of the expansion wave is given by OA and OSB respectively. When the characteristics reach the end of the tube they are reflected back toward the nozzle (AE and BF). The flow conditions in and over the expansion wave may be computed using ”simple wave” theory [51] where:

\[ u_1 + \frac{2a_1}{\gamma - 1} = \frac{2a_0}{\gamma - 1} \]  

holds along a characteristic. The flow velocity \( u_1 \) in the storage tube is obtained from:

\[ \frac{u_1}{a_0} = \frac{M_1}{1 + \frac{\gamma - 1}{2} M_1} \]  

where \( a_0 \) is the speed of sound based on storage tube conditions and \( M_1 \) is the resulting Mach number in the storage tube. Region 0 corresponds to the flow conditions before and region 1 corresponds to conditions behind the expansion wave respectively.

The total temperature ratio is not constant during such a process and the ratio between the storage tube temperature and the total temperature after the expansion wave may be derived as [10]:

\[ \frac{T_{t,1}}{T_0} = \frac{1 + \frac{\gamma - 1}{2} M_1^2}{(1 + \frac{\gamma - 1}{2} M_1)^2} \]  

(2.3)
corresponding to a pressure ratio of:

\[
p_{t,1} \frac{p_{0}}{p_{0}} = \left\{ \frac{1 + \frac{\gamma - 1}{2} M_{1}^{2}}{1 + \frac{\gamma - 1}{2} M_{1}^{2}} \right\}^{\frac{\gamma + 1}{2(\gamma - 1)}}
\]

The total running time of the facility with constant storage tube conditions is governed by the time it takes before the reflected expansion wave reaches the fast acting valve. It may be calculated from (see appendix A for the derivation):

\[
t_{1} = \frac{L}{a_{0}} \cdot \frac{2}{1 + M_{1}} \left( 1 + \frac{\gamma - 1}{2} M_{1}^{2} \right)^{\frac{\gamma + 1}{2(\gamma - 1)}}
\]

The running time \( t_{1} \) is also represented in figure 2.2 by the time it takes for an air particle going from condition 0 to E, where \( L \) is the length of the storage tube. Finally the Mach number of the flow downstream of the expansion wave is determined by imposing mass conservation, it depends directly on the tube diameter to the critical throat diameter (denoted by the superscript *) ratio:

\[
\left( \frac{d_{\text{tube}}}{d^{*}} \right)^{2} = \frac{1}{M_{1}} \left[ \frac{2}{\gamma + 1} \left( 1 + \frac{\gamma - 1}{2} M_{1}^{2} \right) \right]^{\frac{\gamma + 1}{2(\gamma - 1)}}
\]

The impact of free stream Mach number \( M \) through \( d^{*} \) on the variation of total quantities and run time is given in table 2.1. As the test section diameter is fixed, the free stream Mach number is set by selecting the appropriate nozzle diameter. For free stream Mach numbers below 9, a special implementation referred to as a “tandem nozzle” is used (see section 2.2.2). The first (throttle) nozzle in the tandem setup is the Mach 9 nozzle, a second nozzle is then selected for the appropriate lower free stream Mach number. Since the first nozzle determines the stagnation conditions and the run time only the characteristics for Mach numbers 9 and higher are listed in table 2.1. The table shows an increase in losses of total flow quantities and running.
time with decreasing Mach number. However, it is found that in reality the additional total temperature losses at the fast-opening valve due to heat transfer dominate (see section 2.2.3).

To prevent condensation in the free stream, the air in the storage tube is heated. In order to further increase the run time of the tunnel and to reduce the amount of energy required only the section of the storage tube is heated that contains the air that is actually used during the run. To compute this length, the most upstream location of a particle that can reach the valve within the run-time (path DCE in figure 2.2) is considered, yielding approximately:

\[
\Delta L = M_1 \frac{a_1 - u_1}{a_0} 2L
\]  \hspace{1cm} (2.7)

For a total tube length of \( L = 29 \) m (see table 2.2) a distance of \( \Delta L = 5.2 \) m results. Therefore only the first 6 m of the hot tube is heated. In order to reduce the total length of the facility, the storage tube is divided into three sections consisting of the hot tube and cold tube divided into two parts. The hot tube is connected by means of a bended tube to the first cold tube section that is located beneath the hot tube and test section. The second cold tube is located next to and connected to the first tube by a 180° bend, see figure 2.3. Since the cold tube is positioned below the hot tube, the problem of convection in the tube is minimized.

![Figure 2.3: HTFD storage tube layout](image)

The maximum temperature in the storage tube is restricted to a maximum value of 773 K (500°C) by safety regulations for high pressure and high temperature vessels.

<table>
<thead>
<tr>
<th>( M )</th>
<th>( M_1 )</th>
<th>( \frac{u_1}{v_c} )</th>
<th>( \frac{p_1}{p_0} )</th>
<th>( \frac{L_{12}}{L} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>0.09</td>
<td>0.97</td>
<td>0.89</td>
<td>1.94</td>
</tr>
<tr>
<td>10</td>
<td>0.05</td>
<td>0.98</td>
<td>0.93</td>
<td>1.96</td>
</tr>
<tr>
<td>11</td>
<td>0.03</td>
<td>0.99</td>
<td>0.95</td>
<td>1.97</td>
</tr>
</tbody>
</table>

Table 2.1: Expansion wave characteristics
Total length storage tube \( L_{\text{tot}} = 29 \, m \)
Length hot tube \( L_{\text{hot}} = 6 \, m \)
Length cold tube \( L_{\text{cold}} = 23 \, m \)
Diameter test section \( D = 350 \, mm \)
Diameter hot tube \( D_{\text{hot}} = 49.25 \, mm \)
Diameter cold tube \( D_{\text{cold}} = 59 \, mm \)

Table 2.2: HTFD dimensions

<table>
<thead>
<tr>
<th>( M_{\text{perf}} )</th>
<th>( \epsilon_M ) [%]</th>
<th>( \epsilon_u ) [%]</th>
<th>( \epsilon_T ) [%]</th>
<th>( \epsilon_p ) [%]</th>
<th>( \epsilon_\rho ) [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>-0.5</td>
<td>1.1</td>
<td>3.1</td>
<td>1.1</td>
<td>-2.0</td>
</tr>
</tbody>
</table>

Table 2.3: Error in the free stream static quantities when assuming constant \( \gamma \).

For temperatures of this order the specific heat ratio \( \gamma \) is no longer constant since the internal energy stored within the vibrational degree of freedom cannot be neglected anymore. In general the effect of the vibrational mode is modeled by means of a simple harmonic vibrator [86]. For a thermally perfect gas the specific heat ratio is given as a function of the temperature by:

\[
\gamma = 1 + \frac{\gamma_p - 1}{1 + (\gamma_p - 1) \left\{ \left( \frac{\theta}{T} \right)^2 \frac{\exp(\theta/T)}{[\exp(\theta/T) - 1]^2} \right\}}
\]

(2.8)

where for air \( \gamma_p = 1.4 \) and the molecular vibrational-energy constant \( \theta = 3055 \, K \). This relation is plotted in figure 2.4 for a temperature range of 0 to 1000 \( K \). For

\[
\gamma \text{ [\%]} \quad T \text{ [K]}
\]

<table>
<thead>
<tr>
<th>( T )</th>
<th>( \gamma )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1.41</td>
</tr>
<tr>
<td>200</td>
<td>1.40</td>
</tr>
<tr>
<td>400</td>
<td>1.39</td>
</tr>
<tr>
<td>600</td>
<td>1.38</td>
</tr>
<tr>
<td>800</td>
<td>1.37</td>
</tr>
<tr>
<td>1000</td>
<td>1.36</td>
</tr>
</tbody>
</table>

Figure 2.4: Variation of specific heat ratio with temperature

\( T = 773 \, K \) the value for \( \gamma = 1.36 \), which is a relatively small deviation from the constant value of 1.4. In table 2.3 the errors are indicated for the free stream quantities when assuming a constant \( \gamma \). These are small, so the effect of a variable \( \gamma \) is disregarded in the remainder of the investigation.

The discontinuity between the high and low temperature sections of the storage tube may cause a reflection of the expansion wave unless special precautions are taken.
The reflection can be mitigated by changing the tube cross section accordingly. Since the cross sections of the hot and cold part are fixed (see table 2.2), the temperature should be set according to (see appendix B for the derivation):

\[ \frac{d_{\text{hot}}}{d_{\text{cold}}} = \left( \frac{T_{\text{cold}}}{T_{\text{hot}}} \right)^{\frac{1}{2}} \]  

(2.9)

Hence for a hot tube temperature of \( T_{\text{hot}} = 773 \text{ K} \), a cold tube temperature of \( T_{\text{cold}} = 375 \text{ K} \) is needed for a completely reflection free operation.

In figure 2.5 a typical pressure variation with time in the storage tube is given for the M9-M7 tandem nozzle. The pressure transducer is located just upstream of the valve (pressure sensor 1 in figure 2.6). Initially the pressure is equal to the storage tube pressure which in this case is \( p_0 = 99.8 \text{ bar} \), subsequently the pressure drops due to the expansion wave to approximately 83 bar. The sensor is flush mounted with the tube wall therefore the static pressure \( p_1 \) is measured, since the Mach number in the tube is less than 0.1 (see table 2.1) this is practically equal to the total pressure behind the expansion wave \( p_{t,1} \). Typically the rms pressure fluctuation in the storage tube after the expansion wave has passed is 0.3 bar.

After approximately 25 ms a small oscillation is detected, which is caused by a small reflection coming from the temperature/cross section discontinuity. Progressing further in time, a slow pressure rise is observed. This is attributed to the cold air that enters the hot tube and is heated under constant volume.

![Figure 2.5: Pressure variation in time in the hot part of the storage tube and in the settling chamber](image)

2.2.2 Nozzle flow

The HTFD has a nozzle with a conical shape having a 15° total opening angle. The free stream Mach number is varied by changing the throat diameter, various throat sections are available, see table 2.4.
For Mach numbers exceeding 9 the throat section is directly connected to the conical nozzle. For lower Mach numbers an extra throttle nozzle is needed because the throat diameter is limited by the smallest cross section present in the valve. If the throat section is larger, sonic conditions in the throat cannot be reached. The first part of the tandem nozzle system is a Mach 9 nozzle which accelerates the flow to supersonic speeds after which (in the theoretical case) it is decelerated to subsonic speeds through a normal shock wave. The second throat reaccelerates the flow to the desired Mach number, see figure 2.6. Due to the shock wave in the tandem nozzle a total pressure loss occurs, which allows for a larger critical cross section where sonic conditions are achieved. The total pressure loss can be quantified imposing mass conservation and considering its ratio over a normal shock wave:

\[
\frac{p_{t,2}}{p_{t,1}} = \left(\frac{d_1^*}{d_2^*}\right)^2
\]

In practice \(p_{t,1}\) is the total pressure (downstream of the expansion wave) in the storage tube and \(p_{t,2}\) equals the free stream total pressure. The pressure in the settling chamber between the two throats (pressure sensor 2 in figure 2.6) for a free stream Mach number of 7 is measured and it is indicated in figure 2.5. The measured local static pressure is \(25 \pm 1\) bar. The total pressure is obtained using the local Mach number that is calculated from the ratio of the local cross section (diameter 70 mm) with respect to the throat cross section. The local Mach number is 0.44, this in combination with the measured static pressure results in a total pressure of \(p_{t,2} = 28.8 \pm 1\) bar. The measured pressure ratio now becomes \(p_{t,2}/p_{t,1} = 28.8/83.0 = 0.35\). The geometrical area ratio is \((d_1^*/d_2^*)^2 = (19.35/34.3)^2 = 0.32\), this is slightly lower than the measured value. Apparently there is a shock train in the tandem nozzle block that causes less total pressure loss compared to a single normal shock.

**Nozzle boundary layer**  Downstream of the throat the flow expands in the conical nozzle. The length of the nozzle is determined by the throat-to-test-section area ratio and opening angle, for Mach 7 this is 1.18 m. For a one dimensional steady inviscid
flow the area ratio determines the free stream Mach number through the Mach area-ratio relation:

\[
\frac{A}{A^*} = \frac{1}{M} \left( \frac{2}{\gamma + 1} + \frac{\gamma - 1}{\gamma + 1} M^2 \right) \frac{\gamma + 1}{2(\gamma - 1)}
\]  

(2.11)

However, the displacement effect of the nozzle boundary layer can have an appreciable impact by decreasing the area ratio, causing a lower free stream Mach number [53]. Commonly, estimates of the boundary layer thickness are used to correct for these deviations. To determine the boundary layer thickness at the end of the nozzle, PIV measurements were performed for the Mach 7 nozzle. The measurement location is indicated in figure 2.7. Since the optical access to the nozzle was limited, only a single camera could be placed at a relatively large viewing angle with respect to the illuminated plane (> 45°). Although the velocity is less accurate, the relative values still enable to extract a boundary layer profile to infer the integral boundary layer quantities. The measured profile is shown in figure 2.8 with a 7th order polynomial curve fit. The compressible boundary layer integral quantities were obtained using the turbulent Crocco-Busemann relation [92] to obtain the density distribution from the velocity profile:

\[
\frac{\rho_e}{\rho} = \frac{T}{T_e} = 1 + \left( r\frac{\gamma - 1}{2} M_e^2 \right) \left( 1 - \left( \frac{u}{u_e} \right)^2 \right) - \left( 1 + r\frac{\gamma - 1}{2} M_e^2 - \frac{T_w}{T_e} \right) \left( 1 - \frac{u}{u_e} \right)
\]  

(2.12)

where the subscript e denotes the values at the boundary layer edge and \( T_w \) is the nozzle wall temperature.

---

**Figure 2.7: Overview of the nozzle and test section**

A displacement thickness of \( \delta^* = 1 \ mm \) was obtained, the rest of the boundary layer parameters are given in table 2.5. In Lukasiewicz [53] an empirical correlation is given for the variation of displacement thickness with the Reynolds number evaluated at the boundary layer reference temperature:

\[
\frac{\delta^*}{x} = 0.42 R_{ref}^{-0.2775}
\]  

(2.13)
Figure 2.8: Boundary layer profile at the end of the conical Mach-7 nozzle

<table>
<thead>
<tr>
<th>Unit Reynolds number</th>
<th>( Re/m = 14.3 \times 10^6 \text{ m}^{-1} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>95% boundary layer thickness</td>
<td>( \delta_{95} = 13 \text{ mm} )</td>
</tr>
<tr>
<td>Displacement thickness</td>
<td>( \delta^* = 11 \text{ mm} )</td>
</tr>
<tr>
<td>Momentum loss thickness</td>
<td>( \theta = 0.63 \text{ mm} )</td>
</tr>
<tr>
<td>Shape parameter</td>
<td>( H = 18 )</td>
</tr>
</tbody>
</table>

Table 2.5: Boundary layer parameters for the Mach-7 nozzle flow

When the above correlation is applied to the Mach 7 nozzle for \( p_t = 28 \text{ bar} \) and \( T_t = 579 \text{ K} \), a displacement thickness of \( \delta^* = 12 \text{ mm} \) results. This agrees well with the measured value of \( \delta^* = 11 \text{ mm} \).

The boundary layer displacement thickness introduces a reduction in the actual area ratio from 104.1 to 85.8 resulting in a Mach number decrease at the nozzle exit from 7 to 6.7, see table 2.6. This also affects the maximum divergence encountered in the free stream, which decreases from 7.5° to 7.0°. The free stream conditions used in the remainder of this thesis take into account the above mentioned corrections due to viscous effects in the nozzle.

<table>
<thead>
<tr>
<th>Geometrical area ratio</th>
<th>( \left( \frac{A}{A^*} \right)_{\text{geo}} = 104.1 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach number, 1D inviscid</td>
<td>( M_{1D,\text{inviscid}} = 7 )</td>
</tr>
<tr>
<td>Corrected area ratio</td>
<td>( \left( \frac{A}{A^*} \right)_{\text{cor}} = 85.8 )</td>
</tr>
<tr>
<td>Corrected Mach number</td>
<td>( M_{\text{cor}} = 6.7 )</td>
</tr>
</tbody>
</table>

Table 2.6: Impact of the boundary layer on nominal wind tunnel operating conditions
2.2.3 Free stream conditions

When the flow enters the test section it continues diverging due to the outflow from the conical nozzle. Therefore, the flow will expand further and the Mach number in the test section will be higher than obtained from the theoretical area ratio. For the Mach-7 nozzle the theoretical Mach number in the center of the test section is computed from 1D inviscid theory as \( M_{\infty, \text{theory}} = 7.7 \). After correcting for the boundary layer displacement thickness this is \( M_{\infty, \text{cor}} = 7.5 \). The local Mach number variation due to the diverging flow field in the center of the test section may be calculated by:

\[
\frac{dM}{dx} = \frac{2M \left( 1 + \frac{2-1}{2} M^2 \right) \tan \phi}{M^2 - 1} \frac{\tan \phi}{r}
\]

(2.14)

where \( \phi \) is the nozzle semi aperture angle (corrected for the boundary layer displacement effect) and \( r \) is the test section radius which is obtained from the Mach area-ratio relation (equation (2.11)). For \( M = 7.5 \) and \( \phi = 7^\circ \), \( \frac{dM}{dx} = 0.023 \text{ cm}^{-1} \). For a typical model having a length of \( L = 10 \text{ cm} \) a \( \Delta M = 0.23 \) results.

Since the test section has a cylindrical shape, a shock wave is formed at the nozzle test-section junction, see figure 2.7. For \( M = 6.7 \) (Mach number at the location of the nozzle test-section junction) and a compression angle of \( \phi = 7^\circ \) a shock with an angle of \( \theta = 14^\circ \) with respect to the free stream is formed, the shock angle with respect to the test section wall is \( 7^\circ \).

In figure 2.9 the vertical component of the free stream velocity field in the test section is shown. The free stream part of the flow was measured using conventional PIV at the location shown in figure 2.7. In order to visualize the junction shock the camera was oriented under an angle with respect to the illuminated plane requiring a stereo PIV setup.

The solid black circle in figure 2.9 shows the outline of the windows. By plotting the vertical flow component the junction shocks are clearly visualized. The approximate angles are \( 9^\circ \) and \( 10^\circ \) at the bottom and top respectively. The difference between the measured and theoretical shock angle is due to the presence of a cavity at the bottom for the optical access and a larger cavity at the top accommodating the model support system. In addition, the observed difference between shock angles at bottom and top of the test section is due to the different cavity sizes. However this has no influence on the free stream flow due to the hyperbolic nature of the supersonic/hypersonic regime. In the center of the test section a region of \( 200 \times 200 \text{ mm}^2 \) of undisturbed flow is attained.

Figure 2.9 also clearly shows the flow divergence caused by the conical nozzle. A profile of the vertical flow component in the center of the test section is plotted in figure 2.10. The vertical component varies from \( -60 \) to \( +60 \text{ m/s} \) over a distance of \( 200 \text{ mm} \). This agrees well with the theoretical divergence corresponding to a total opening angle of \( 14^\circ \) indicated in figure 2.10.

A profile of the mean velocity magnitude in the center of the test section is given in figure 2.11. The mean velocity is \( 1033 \text{ m/s} \) and the data deviation from uniform flow is 0.2\% (rms fluctuations within a single velocity field) and a repeatability of 0.4\% (rms fluctuation with different realizations). Figure 2.12 shows the free stream velocity along the centerline of the test section, the increase in velocity with downstream distance is caused by the diverging flow.
Figure 2.9: Vertical velocity component of the free stream flow for the Mach 7 nozzle

Figure 2.10: Flow divergence in the test section, vertical velocity component

Figure 2.11: Vertical profile at \( x = 0 \) of the free stream velocity for the Mach 7 nozzle

Figure 2.12: Horizontal profile at \( y = 0 \) of the free stream velocity for the Mach 7 nozzle

**Total temperature** From the free stream velocity and Mach number it is possible to obtain the static and total temperature in the flow. In hypersonic short duration facilities these two quantities are notoriously difficult to determine [31]. However, from the definition of the Mach number and the measured velocity, the static temperature can be directly determined:

\[
T = \left( \frac{|V|}{MT} \right)^2 \frac{\gamma R}{\gamma R} \tag{2.15}
\]
Velocity | $|V| = 1033 \text{ m/s}$
Mach | $M = 7.5$
Static temperature | $T = 47 \text{ K}$
Total temperature | $T_t = 579 \text{ K}$
Total enthalpy | $H_0 = 0.56 \text{ MJ/Kg}$

Table 2.7: Free stream conditions for the Mach 7 nozzle configuration

The total temperature can now be calculated from the energy equation:

$$T_t = T + \frac{|V|^2}{2c_p}$$  \hspace{1cm} (2.16)

The resulting static and total temperature are given in table 2.7. For the determination of the temperature the average velocity in the center of the test section is taken.

A discrepancy is observed between the total temperature obtained from equations 2.15 and 2.16 and the storage temperature. In figure 2.13 the measured free stream velocity is given by the solid line as a function of the storage tube temperature, the dotted line corresponds to the free stream velocity for a fully adiabatic flow where the total temperature loss over the expansion wave (equation 2.3) is taken into account. There is a 10% difference between the measured and theoretical velocity. It is thought that losses because of heat transfer in the valve and tandem nozzle are responsible for this.

Figure 2.13: Free stream velocity as a function of the storage tube temperature

**Pressure measurements** The free stream Mach number and total pressure are alternatively evaluated by means of pressure measurements. Static pressure measurements in the nozzle are performed in combination with Pitot tube measurements in the test section, see figure 2.7. The Pitot tube formula correlates the measured Pitot pressure $p_{t,p}$ to the total free stream pressure $p_{t,1}$ and the local Mach number $M_p$ (at
the position of the Pitot tube):

\[
\frac{p_{t,P}}{p_{t,1}} = \left[ \frac{\gamma + 1}{2\gamma M_p^2 - (\gamma - 1)} \right]^\frac{\gamma-1}{\gamma-1} \left[ \frac{(\gamma + 1) M_p^2}{(\gamma - 1) M_p^2 + 2} \right]^\frac{\gamma}{\gamma-1}
\]

(2.17)

The static pressure measurements \( p_s \) also correlate the total pressure (which is considered to be constant throughout the nozzle and test section) and the local Mach number \( M_s \):

\[
\frac{p_{t,1}}{p_s} = \left( 1 + \frac{\gamma - 1}{2} M_s^2 \right) ^\frac{\gamma}{\gamma-1}
\]

(2.18)

To be able to solve the two equations it is assumed that the flow divergence is equal to the inviscid case corrected for the boundary layer displacement thickness. Now, the local Mach number at the location of the static pressure orifices can be related to the Mach number at the Pitot tubes using the nozzle opening angle and the absolute distance \( \Delta x \) between the static pressure orifices and Pitot tubes. From geometrical relations it follows that:

\[
\frac{A_s}{A^*} = \left( \frac{r^* \sqrt{\frac{A_s}{A^*} - \Delta x \tan \phi}}{r^*} \right)^2
\]

(2.19)

where the local Mach number is related to the local cross section by the Mach area-ratio relation (see equation 2.11). Finally the equations are solved for \( p_{t,1} \) and \( M_p \).

![Figure 2.14: Static and Pitot pressure signal during a wind tunnel run](image)

In the measurements static pressure taps were installed at \([95, 295, 495] \text{ mm}\) upstream of the nozzle test section junction, see figure 2.7. Simultaneously, a Pitot probe was installed in the test section, it is located 300 \( \text{ mm} \) downstream of the junction, so that \( \Delta x = [395, 595, 795] \text{ mm}\). In the current procedure the three static pressure
measurements are used in combination with two Pitot pressure signals in the optimization for finding \( p_t \) and \( M_p \). For the pressure signals given in figure 2.14 the following average values are obtained: \( M_p = 7.5 \pm 0.1 \) and \( p_t = 27.9 \pm 1 \) bar (also see table 2.8).

### 2.2.4 Stagnation heat flux measurements

In order to determine the free stream temperature for the nozzle setup without the tandem nozzle configuration, the stagnation heat flux on a 50 mm radius sphere-cylinder was measured by means of quantitative infrared thermography. Due to the presence of seeding particles in the flow, condensation occurred for the M9 nozzle which prevents to determine the static and total temperature by means of PIV. In figure 2.15 the stagnation heat flux variation with time is shown for different Reynolds numbers (total pressures). The horizontal lines in the figure are the theoretical values obtained using the Fay-Riddell stagnation point heat transfer correlation [32]:

\[
q = 0.94 \left( \rho_c \mu_c \right)^{0.4} \left( \rho_w \mu_w \right)^{0.1} \sqrt{\frac{\text{d}u_e}{\text{d}x}} (h_{aw} - h_w)
\]  

(2.20)

The temperature difference used to compute \( c_h \) from the measured and calculated heat flux in figure 2.15 is \( T_I - T_w \) since the adiabatic wall temperature (see section 4.4.5) is difficult to obtain accurately within such short run times.

| \( p_{s,1} \) [bar] | \( p_{s,2} \) [bar] | \( p_{s,3} \) [bar] | \( p_{p,1} \) [bar] | \( p_{p,2} \) [bar] | \( M \) | \( p_t \) [bar] |
|---------------------|---------------------|---------------------|---------------------|---------------------|---------------------|
| 0.01                | 0.0156              | 0.0273              | 0.3123              | 0.3197              | 7.5                 | 27.9                |

Table 2.8: Average measured pressure with the resulting Mach number and total pressure.
<table>
<thead>
<tr>
<th>Variable</th>
<th>Measurement method</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach number $M$</td>
<td>Theory with BL correction, Static-Pitot pressure measurement</td>
<td>$7.5 \pm 0.1$</td>
</tr>
<tr>
<td>Total pressure $p_t$ [bar]</td>
<td>Pressure upstream of the throat, Static-Pitot pressure measurement</td>
<td>$28.6 \pm 1$</td>
</tr>
<tr>
<td>Total temperature $T_t$ [$K$]</td>
<td>Velocity by means of PIV in M7 nozzle, Fay-Riddell evaluation in M9 nozzle</td>
<td>579, 585</td>
</tr>
</tbody>
</table>

Table 2.9: Overview of methods to assess the free stream characteristics of the HTFD at a storage pressure of 100 bar and storage temperature of 773 K

The total temperature was varied to fit the results from the Fay-Riddell correlation to the measured heat flux. The best agreement was found for $T_t = 585$ K, which corresponds well to the total temperature for the M7 nozzle (see previous section).

### 2.2.5 Operational envelope

The wind tunnel was extensively operated with the Mach 7 nozzle at a storage tube pressure of 100 bar and a storage temperature of 773 K. The displacement effect of the boundary layer was measured and the effect on the free stream Mach number was evaluated. Simultaneously, the Mach number was obtained from static and Pitot tube measurements (section 2.2.2). Both evaluations agreed on a test section Mach number of $M = 7.5$. The total pressure was inferred from the static-Pitot pressure measurements and the pressure measured just upstream of the second throat, the result was a total pressure of $p_t = 27.9$ bar (section 2.2.2). The free stream velocity was used to calculate the free stream total temperature $T_t = 579$ K. In case of the Mach 9 nozzle a total temperature of $T_t = 585$ was obtained by means of Fay-Riddell evaluations. For an overview of the methods used to determine the free stream variables, see table 2.9.

The free stream characteristics for the Mach 7 nozzle are used to establish the approximate Mach-Reynolds operational envelope of the HTFD. The free stream Mach number is obtained from the theoretical area ratio corrected for the displacement effect of the nozzle boundary layer. To obtain the displacement thickness for other Mach numbers and total pressures, the correlation from equation 2.13 is used in combination with the conditions in the Mach 7 nozzle as a reference point:

$$\frac{\delta^*}{(\delta^*)_{M7}} = \left( \frac{Re_\text{ref}}{(Re_\text{ref})_{M7}} \right)^{-0.2775}$$  \hspace{1cm} (2.21)

The maximum total pressure is determined by the maximum allowable pressure in the storage tube and the losses when the tandem nozzle is installed. The minimum pressure is determined by the pressure ratio needed to operate the tunnel at the given Mach number. The total temperature is 585 K for nozzle configurations without the throttle nozzle and 579 K with the throttle nozzle installed.

The free stream total quantities as well as Mach and unit Reynolds number for different nozzle geometries are given in table 2.10 and in figure 2.16. The total temperature is fixed at the maximum to prevent condensation issues, however for the lower
Mach numbers (M6 and M7 nozzles) the temperature could be reduced to achieve a higher Reynolds number. For the evaluation of the unit Reynolds number, the viscosity is calculated using Sutherland’s law [4]. As can be observed the unit Reynolds number increases with the free stream Mach number up to the Mach 9 nozzle, this is because the pressure loss in the tandem nozzle decreases. For the Mach 10 nozzle, the unit Reynolds number decreases as expected from theory.

<table>
<thead>
<tr>
<th>Nozzle</th>
<th>$M$</th>
<th>$p_t$ [bar]</th>
<th>$T_t$ [K]</th>
<th>$Re/m \times 10^6$ [m$^{-1}$]</th>
</tr>
</thead>
<tbody>
<tr>
<td>M6</td>
<td>6.4</td>
<td>2.8</td>
<td>579</td>
<td>1.61</td>
</tr>
<tr>
<td></td>
<td>6.5</td>
<td>14.3</td>
<td>579</td>
<td>7.90</td>
</tr>
<tr>
<td>M7</td>
<td>7.4</td>
<td>5.4</td>
<td>579</td>
<td>2.22</td>
</tr>
<tr>
<td></td>
<td>7.5</td>
<td>28.0</td>
<td>579</td>
<td>11.05</td>
</tr>
<tr>
<td>M8</td>
<td>8.4</td>
<td>10.0</td>
<td>579</td>
<td>3.07</td>
</tr>
<tr>
<td></td>
<td>8.5</td>
<td>51.2</td>
<td>579</td>
<td>15.08</td>
</tr>
<tr>
<td>M9</td>
<td>9.4</td>
<td>20</td>
<td>585</td>
<td>4.65</td>
</tr>
<tr>
<td></td>
<td>9.5</td>
<td>88</td>
<td>585</td>
<td>19.70</td>
</tr>
<tr>
<td>M10</td>
<td>10.3</td>
<td>20</td>
<td>585</td>
<td>3.76</td>
</tr>
<tr>
<td></td>
<td>10.5</td>
<td>88</td>
<td>585</td>
<td>15.85</td>
</tr>
</tbody>
</table>

Table 2.10: Free stream total quantities for different nozzles and total pressures

Since the free stream static temperature is relatively low it should be carefully verified that the flow is not affected by condensation. In general two types of condensation processes are possible: heterogeneous condensation that behaves like equilibrium con-

2.2.6 Condensation effects

Figure 2.16: Operational Mach Reynolds envelope for a total temperature of $T_t = 555$ K
densation and spontaneous condensation for which supercooling is possible. Heterogeneous condensation is characterized by the process where the degree of CO$_2$ and H$_2$O content in air is sufficient to cause condensation of nitrogen and oxygen around the existing nuclei. Spontaneous condensation is characterized by the nucleation of nitrogen molecules which then act as nuclei for the rest of the nitrogen and oxygen. The spontaneous condensation process is governed by the ratio of expansion rate to static pressure, $\frac{\dot{P}}{p}$ [19], which indicates the possible amount of supercooling. For the HTFD free stream this quantity is given by:

$$\frac{\dot{P}}{p} = \frac{\sqrt{\gamma RT_t}}{p_t} \cdot \frac{2\gamma M^4}{M^2 - 1} \cdot r \tan \phi$$

(2.22)

where $\dot{P} = -(1/p)(dp/dt)$, $r$ is the local radius and $r^*$ is the throat radius. In case $\frac{\dot{P}}{p}$ is less than about 0.1 $s^{-1} mbar^{-1}$ little or no supercooling is expected since the process is dominated by heterogeneous condensation. For HTFD the value of $\frac{\dot{P}}{p}$ is around $3 - 50 s^{-1} (mbar)^{-1}$ where the high values are obtained for the higher Mach numbers. From this it can be concluded that the condensation process is dominated by spontaneous condensation and supersaturated conditions can be achieved. In figure 2.17 a pressure-temperature diagram is given in which data is depicted for condensation onset in hypersonic facilities [19]. In the figure also the mean onset of nitrogen as obtained from experiments is shown in green and the equilibrium condensation line in red. When these lines are crossed from the lower right-hand corner to the upper-left, condensation occurs for a supercooled flow and an equilibrium flow respectively. In general the expansion rate decreases with increasing nozzle size, therefore, the amount of supercooling for large facilities is reduced resulting in an earlier condensation onset [53]. For low static pressures ($\approx 1 mbar$) the seeding effect of CO$_2$ and H$_2$O is negligible and spontaneous condensation dominates the process. The black dots indicate the HTFD static conditions at different Mach numbers. For the M7 nozzle the static conditions are still within the equilibrium condensation limits for N$_2$. At higher Mach numbers static temperatures are achieved for which equilibrium condensation would occur. However, previous experience with the M9 nozzle shows that also this configuration achieves a freestream without condensation. Additionally, it may be remarked that the Purdue and Braunschweig facilities operate under similar relatively low stagnation conditions where also supercooling has been observed.

### 2.3 Implementation of measurement techniques

The wind tunnel is made such that it is able to generate the desired flow conditions. However the measurement techniques that are used also require specific infrastructure such as a facility to apply particle seeding in the flow for particle image velocimetry, sufficient optical access and proper synchronization of the wind tunnel with the measurement technique. Especially the latter is important when working in short duration facilities.
Figure 2.17: Experimental estimates of condensation onset [19, 53] (‘st’ means small tunnel $d_{\text{test section}} < 30 \text{ cm}$ and ‘lt’ means large tunnel $d_{\text{test section}} > 30 \text{ cm}$)

### 2.3.1 Particle Image Velocimetry (PIV)

In order to facilitate particle image velocimetry in the HTFD, a number of aspects are required. First of all tracer particles have to be added to the flow and optical access has to be provided to enable illumination and recording.

The tracer particles are introduced in the hot storage tube off-line by means of a high-pressure cyclone device. The inlet is located in the upstream proximity of the fast acting valve. The amount of seeding that is introduced requires careful control, which is achieved by introducing the seeded airflow by a finite amount through successive charge-discharge of the seeder chamber fluid into the hot storage tube. However, due to the single inlet port a non-uniform particle distribution is obtained although there is some transport into the tube by the inflow and some expected thermal convection. Consequently the seeding density distribution decreases during the run time of the tunnel. The PIV hardware initially used in the experiments has a maximum recording frequency of 3.3 $Hz$; hence only a single snapshot can be acquired during a run. The seeding level is therefore optimized for a given time instant (30 $ms$ after valve opening) selected for the measurement. The optimum is experimentally determined, enabling the system to produce acceptable control and repeatability of the seeding conditions.

A schematic of PIV illumination and imaging setup is given in figure 2.19. In the central bottom part of the test section a window with a diameter of 150 $mm$ is provided that allows to illuminate the model by a laser. The side windows are used as optical access for the cameras and have a 280 $mm$ diameter. Typically a field of view (FOV) ranging from $4 \times 3$ to $8 \times 6 \text{ cm}^2$ is used. The laser pulse time separation $\Delta t$ ranges from 0.6 to 3 $\mu s$ essentially depending on the optical magnification.

The PIV system is synchronized with the wind tunnel operation such that it can
perform a measurement at any selected time instant during the run (see section 2.3.4). In figure 2.20 a sample PIV result is given for the flow over the aft part of a double compression ramp, the total flow field picture is a combination of different fields of view.

2.3.2 Quantitative Infrared Thermography (QIRT)

The HTFD can be fitted with a Germanium window that provides optical access in the infrared wavelength range. The window has a diameter of 200 mm and is located in the center of the normal optical access ports through an adapter flange. The transmissivity of the Germanium window is included in the calibration of the IR camera, see section 4.3.3. The model is viewed under an angle to prevent the camera to view reflections of itself and the cold detector in the Germanium window and wind tunnel model (narcissus effect).
To optimize the performance of IR thermography, some requirements can be formulated regarding the wind tunnel model material. It should have a low thermal product \((\rho c d)\) in order to maximize the surface temperature increase for a given heat flux. Furthermore, the conductivity should be low so that lateral conduction is limited as much as possible. To further maximize the signal received by the infrared camera, the emissivity should be close to 1. The wind tunnel model material used for QIRT measurements is Makrolon. This is a low conductive \((k = 0.20 \text{ W}/\text{Km})\) polycarbonate material that allows for accurate manufacturing due to a high creep modulus and isotropic behavior. The surface emissivity was measured to be \(\epsilon = 0.88\) ([75]). Moreover, Makrolon can withstand temperatures as high as 120 °C with unaltered properties, which is below the maximum expected temperature during the thermal transient.

2.3.3 Schlieren

The setup of the schlieren system is shown in figure 2.23. It features two parabolic mirrors to provide a collimated beam and focusing. The schlieren filter is a knife edge that can be rotated in the plane perpendicular to the light beam. Illumination is either done by a continuous light source or a xenon spark source (20 ns emission). Typically a pinhole size of approximately 1 mm is used. A typical schlieren visualization is given in figure 2.24.

2.3.4 Synchronization

Since the flow duration of the facility is limited to approximately 100 ms, a synchronization system was developed that enabled the usage of low repetition rate cameras. The system is capable of working in master and slave mode. In the master mode the wind tunnel is the master and triggers the measurement system. A TTL (0–5 V) trigger signal is based on the pressure drop when the valve is opened. However, in some cases another approach is desired where the wind tunnel is slave. Here a pulse-train
is supplied to the facility, a computer program determines the pulse time separation and computes the total delay (phase shift) that is needed for the requested timing. The total delay is obtained from the pneumatic delay and the user input delay. Generally the system is used in slave mode, where in case of PIV the laser acts as the pulse generator. When applying spark schlieren, a pulse generator is used to control the wind tunnel and flash lamp/camera. For the PIV experiments the time delay was set to $30\, ms$ into the run with a standard deviation of $5\, ms$. In figure 2.25 an example is given where the trigger pulse is given in combination with the storage tube pressure signal. The width of the trigger pulse is $30\, ms$ to assure detectability by the HP1000 data acquisition and processing computer.

Figure 2.23: Setup of the schlieren system

Figure 2.24: Sample image of a schlieren visualization (continuous illumination)

Figure 2.25: Example of storage tube pressure signal and trigger signal for the wind tunnel in slave mode
Chapter 3

Particle Image Velocimetry

3.1 Introduction

Particle image velocimetry is a non-intrusive technique to obtain the velocity field by means of tracer particles. It is a non-intrusive planar technique as opposed to probe measurements (HWA and 5-hole probes). Depending on the application type, the measurement technique returns the in-plane two component (2-C) velocity vector, or in case it is applied in stereoscopic configuration three velocity components (3-C) are returned. Recently the extension to measurement in a volume has been made using a tomographic configuration [28].
In this chapter the measurement principle of PIV and the data processing procedure is discussed. Furthermore, the important effect of particle slip is investigated. A procedure is discussed that enables to experimentally obtain the extent of the particle slip and a correction procedure is introduced.

3.2 Measurement principle and data processing

In figure 3.1 the schematic of a typical PIV setup is given. The steps in performing a PIV experiment are as follows:

**Flow seeding** Particle tracers are introduced in the flow that follow the flow field velocity.

**Illumination** The suspended particles are illuminated twice by a pulsed light source within a thin region of space that is called the measurement plane.

**Recording** The light scattered by the illuminated particles is recorded by means of a frame-straddling CCD camera, which is capable of obtaining image pairs (image A and B) with minimum time separation in the order of 600 ns.

**Data processing** The recorded images are stored and analyzed by means of spatial cross-correlation to evaluate the particle motion.

The following section discusses in detail the above mentioned elements of the work-
Figure 3.1: Configuration of a PIV experiment

The selection of a suitable particle type is an assessment of light scattering capability, particle response and handling characteristics which dictate conflicting requirements. The light scattering capability of a particle is determined by the ratio of the particle refractive index to the surrounding refractive index, the particle size, shape and orientation. Furthermore also the polarization and observation angles have an important influence [63]. When the particle size is larger than the wavelength of the incident light \( \lambda \), Mie scattering occurs. When the particle size is smaller than \( \lambda \), Rayleigh scattering occurs. For Mie scattering the average intensity increases with \( d_p^2 \) where \( d_p \) is the particle diameter, in case of Rayleigh scattering the intensity varies with \( d_p^6 \). From this it becomes evident that the particle scattering capability decreases strongly with the particle diameter, especially when the particles become so small that Rayleigh scattering occurs. So within the limits between Mie and Rayleigh scattering a particle diameters as large as possible is desired.

This however contrasts with the requirement that a particle has to faithfully follow the flow. This is especially important in high speed flow where large accelerations occur (e.g. shock waves and expansion waves). The particle fidelity can be quantified through the relaxation time \( \tau \) (see section 3.3.1):

\[
\tau = \frac{4\rho_p d_p^2}{3c_d Re_d \mu_f}
\]  

where \( \rho_p \) is the particle density, \( c_d \) is the particle drag coefficient, \( Re_d \) is the particle Reynolds number and \( \mu_f \) is the flow viscosity. When the relaxation time is high, the particle fidelity is low. This shows that in order to minimize the relaxation time, the
The particles used in the present experiments are solid titanium dioxide (TiO$_2$) particles with a nominal crystal size of 50 nm and a bulk density of $\rho_p = 200 \, \text{kg/m}^3$, which are chosen because they can easily withstand heating up to 800 K in the HTFD storage tube. Furthermore TiO$_2$ is generally available from the industry in very fine crystal sizes since it is used as a raw material for a wide selection of products (e.g. paint and toothpaste).

In figure 3.2 a scanning electron microscopy image is shown of particles as they are obtained from the powder container. A qualitative inspection shows that the crystal size indeed corresponds to the nominal one (measuring the individual blob size in figure 3.2(a)). However the particles tend to cluster forming agglomerates of approximately 400 nm.

Furthermore a shock wave test was carried out (see section 3.3.2) to assess the actual particle performance. From the test a relaxation time of $\tau = 2.5 \, \mu\text{s}$ resulted [76]. The relaxation time can also be calculated using expression (3.1) where the particle drag coefficient is calculated by the method described in Tedeschi at al. [88]. For a particle diameter $d_p = 400 \, \text{nm}$, a relative Mach number of $\Delta M = 0.7$, relative Reynolds number $Re_d = 4.8$ and a Knudsen number based on the particle diameter of $Kn = 1.2$ a particle drag coefficient of $c_d = 2.2$ results. Applying these values to equation (3.1) results in a particle relaxation time of $\tau = 1 \, \mu\text{s}$. This result is in fair agreement with that obtained experimentally considering the relatively high uncertainty in determining both effective particle diameter (shape) and bulk density (porosity).

### 3.2.2 Particle imaging and illumination

The particle pattern recorded by the camera is diffraction limited which means that the imaged particle diameter $d_{\tau}$ is determined by the optical setup [2]:

$$d_{\tau} = \sqrt{(Md_p)^2 + d_d^2} \quad (3.2)$$
where $M$ is the magnification and $d_d$ is the diffraction spot diameter:

$$d_d = 2.44f\#(M + 1)\lambda$$  \hspace{1cm} (3.3)

In the experiments a particle image of approximately 2 pixels was set by changing the $f\#$, which is the ratio between the focal length of the camera and its aperture. This was done in order to improve the accuracy of the measurement technique and to prevent peak locking [63]. The illumination was provided by a double cavity Nd-YAG laser, depending on the application a Spectra-Physics Quantra-Ray 400 mJ laser was used or a Quantel 200 mJ laser. In both cases ample light was available for illumination.

Figure 3.3 reports a particle image recording to illustrate typical seeding conditions encountered in the experiments. The free-stream flow is uniformly seeded and the compression shocks are visualized as interfaces between the low-density free stream and the denser flow behind the shock (see chapter 5 for a more detailed flow field description). Under the assumption of two-dimensional adiabatic steady flow, the flow density increase can be easily derived from the shock wave equations. The largest density ratio occurs over a normal shock wave, which at Mach 7.5 has a value of 5.5. The same increase is expected in terms of particle seeding density downstream of the particle relaxation region (excluding particle clustering or decomposition phenomena). From the inspection of a single PIV recording, the particle image density increases weakly across the first oblique shock wave that is barely visible along the model surface (normal Mach number 2.6, density ratio 3.4). The increase is more evident across the curved shock emanating from the second compression ramp where the flow exhibits a locally normal shock at a Mach number of 7.5. Large-scale fluctuations and turbulent structures are also visualized in the flow region downstream of the curved shock.

![Particle image recording of the flow over a double compression ramp (two exposures superimposed) at Mach 7.5](image)

Figure 3.3: Particle image recording of the flow over a double compression ramp (two exposures superimposed) at Mach 7.5
The amount of light scattered by the sub-micron diameter particles is shortly assessed by inspecting the pixel intensity histograms. Two conditions are chosen, first the free-stream flow with relatively low seeding density and secondly the flow past the curved shock where it is strongly compressed. In both cases background subtraction is applied by means of a sliding minimum. The histogram is given in figure 3.4 where the red bars represent the imaged free stream and the blue bars represent the area downstream of the curved shock. Both histograms are obtained using the same amount of pixels. For the free stream it is found that 80% of histogram is within 100 levels. This corresponds to approximately a 7-bit intensity range therefore negligible loss of accuracy is expected due to poor quantization [63]. For the flow downstream of the curved shock the histogram is broader corresponding to an 8 to 9 bits range.

![Histogram of pixel intensity in the free stream and downstream of the curved shock including background subtraction](image)

Figure 3.4: Histogram of pixel intensity in the free stream and downstream of the curved shock including background subtraction

### 3.2.3 Data processing

Once the particle images are recorded, the particle displacement is obtained by means of cross correlation. The image is divided in sub-regions, called interrogation regions that have a typical size of 32 × 32 pixels. An interrogation region is selected from the recorded image and the normalized cross correlation function [94] is computed for image A and B:

\[
R(\Delta x, \Delta y) = \frac{\sum_{x,y}^W (I_a(x,y) - \bar{I}_a) (I_b(x+\Delta x, y+\Delta y) - \bar{I}_b)}{\sqrt{\sum_{x,y}^W (I_a(x,y) - \bar{I}_a)^2 \sum_{x,y}^W (I_b(x,y) - \bar{I}_b)^2}}
\]  \hspace{1cm} (3.4)

where \(I_a\) and \(I_b\) are the pixel intensities at locations \(x\) and \(y\), \(W\) is the cross correlation window size, \(\bar{I}_a\) and \(\bar{I}_b\) are the average image intensity within the window and \(\Delta x\) and \(\Delta y\) is the image shift. In figure 3.5 an example is given where \(I_a\) (green) and \(I_b\) (red) are depicted in the same frame. From the image the particle displacement is already apparent. The result of the cross correlation process is shown in figure 3.6 where the position of the peak in the correlation space corresponds with the mean particle displacement \((\Delta \bar{x})\). The particle velocity can be computed from the mean
particle displacement as:

\[ \vec{v} = M \frac{\Delta \vec{t}}{\Delta t} \]  

(3.5)

where \( M \) is the magnification and \( \Delta t \) is the laser pulse time separation. Subsequently a second interrogation region is selected and the process is repeated. This is done until the complete image is analyzed.

Figure 3.5: Two superimposed exposures within an interrogation window. First exposure is red and second exposure is green

Figure 3.6: Correlation map

In order to improve accuracy and robustness an iterative approach is used. First, a multi-grid analysis is applied where the interrogation window size is progressively decreased [85]. This process eliminates the \( \frac{1}{4} \) rule constraint [63] and it is usually terminated when the final window size is reached. Hereafter iterative analysis is performed at a fixed sampling rate (grid spacing) and window size. This process allows to further improve the accuracy of the image deformation ([70]; [60]; [87]) and to a certain extent allows to enhance the spatial resolution of the measurement [74]. Furthermore in the iterative loop image deformation (see figure 3.7) is applied in order to minimize errors due to in-plane velocity gradients.

Figure 3.7: Image deformation according to the local velocity field, taken from [70]

The iterative equation of the PIV process can be described as:

\[ \vec{V}^{k+1} = \vec{V}^k + \vec{C}^{k+1} \]  

(3.6)

Where \( \vec{V}^k \) indicates the result of the evaluation at the \( k^{th} \) iteration and is used as predictor for the following iteration. The correction term \( \vec{C}^k = (c_u, c_v) \) is the vector
determined by cross-correlating the deformed images (equation 3.4): \( \tilde{C}^k = I_a^k \otimes I_b^k \).

The deformed images are obtained from the original images \( I_a^k \) and \( I_b^k \) according to:

\[
I_a^k(x, y) = I_a \left( x - \frac{u_d^k}{2}, y - \frac{v_d^k}{2} \right) \\
I_b^k(x, y) = I_b \left( x + \frac{u_d^k}{2}, y + \frac{v_d^k}{2} \right)
\]  

(3.7)

Here \((u_d, v_d)\) is the image deformation displacement at location \((x, y)\) commonly obtained interpolating the predictor displacement \(V_k\) onto a pixel grid. The whole image interrogation process is described schematically in figure 3.8.

**Figure 3.8:** Block diagram of the iterative image deformation interrogation method

The most common choice for the displacement interpolation is bi-linear ([41], [43], [70]) which is also used in the current applications. Higher order functions have also been considered ([33]; [50]), which may in principle improve the accuracy of image deformation. In all cases where a sub-pixel image transformation is performed (sub-pixel window shift and window deformation), the images have to be interpolated for re-sampling at non-integer pixel positions. The topic is extensively discussed by Astarita [7], who concluded that high-order interpolators are necessary to achieve higher accuracy. However, high noise levels may reduce the relative improvements obtained for high order interpolators [6]. In the current applications image interpolation is performed using a sinc interpolation with a \(11 \times 11\) pixel kernel size. Furthermore filtering is applied in order to prevent the iterative process to become unstable [74]. This is done by applying a two-dimensional second order regression filter to the displacement field. It has been shown that this filter has a favorable combination of noise reduction behavior and low spatial modulation.

### 3.2.4 Stereoscopic PIV

Stereoscopic PIV is used in order to obtain the three component velocity vector field in the measurement plane. As the name indicates, the measurement principle relies on a stereoscopic approach. Two cameras are placed at different angles with respect to the measurement plane where each camera registers the particle displacement perpendicular to its optical axis. From the two velocity vector projections the actual vector can be reconstructed (see figure 3.9).

For a proper reconstruction, the camera viewing direction with respect to the illumination plane should be known. This is achieved by performing a calibration procedure that results in a mapping function allowing to map the projected velocity vector back onto the physical measurement plane, thus obtaining the out-of-plane velocity.
component. Finally a self calibration procedure [93] is applied that uses the particle image recordings to correct for any final errors in the calibration procedure (including misalignment of the calibration plate and light sheet).

The errors in the out-of-plane velocity component are a function of the camera viewing angle. For a viewing angle of 45 degrees the out-of-plane velocity error is minimal and equal to the in-plane error [62]. In the current applications the camera angle is smaller than 45 degrees resulting in an out-of-plane error that will be approximately 4 times the in-plane error.

![Stereoscopic PIV principle](www.lavision.com)

Figure 3.9: Stereoscopic PIV principle [www.lavision.com]

### 3.3 Particle characterisation

The application of PIV fundamentally depends on the requirement that the tracer particles accurately follow the flow field. Due to large flow accelerations present in compressible flows in the form of shock or expansion waves the inertial forces cause particle slip, which reduces the accuracy of the measurement technique in these areas. The amount of particle slip, determined by the particle relaxation time depends on the particle characteristics and flow field. When the amount of particle slip is known the extent of the affected regions can be established and a possible correction strategy may be developed. Theoretical relations can be established that enable to compute the particle relaxation time for given particle properties (density, size) and flow field characteristics (relative Reynolds number, relative Mach number, amount of rarefaction) [57], [88]. However in practice it is found that the uncertainty in particle characteristics (bulk density, effective size due to agglomeration) causes large deviations from the theoretical relaxation values [76] and that an experimental determination of the particle response is required.
3.3.1 Particle response

In order to assess the amount of particle slip, first the behavior of heavy particles in a gas flow is studied. The equation of motion for a particle in a (unsteady) gas flow is given by [57]:

$$\frac{\pi d_p^3}{6 \rho_p} \frac{d\vec{U}_p}{dt} = -3\pi \mu d_p \left( \vec{U} - \vec{U}_p \right) + \frac{\pi d_p^3}{6 \rho} \frac{d\vec{U}}{dt} - \frac{1}{2} \frac{\pi d_p^3}{6 \rho} \frac{d}{dt} \left( \vec{U}_p - \vec{U} \right) - \frac{3}{2} \frac{d_p^2}{\sqrt{\pi \rho \mu}} \int_t^{t_0} \frac{d\left( \vec{U}_p - \vec{U} \right)}{dt'} \sqrt{t - t'}$$

(3.8)

The left-hand side of equation 3.8 represents the product of mass and acceleration. The right-hand side represents the forces acting on the particle. The first two right-hand terms represent the viscous force which will be discussed in more detail and the acceleration force according to Stokes’ law. The third term gives the force needed to accelerate the added mass which, for spherical particles, equals one-half of the mass of the displaced fluid. The last term is known as the Basset force which accounts for the deviation of the flow pattern around the particle from that for steady flow (effect of history of the motion). Since the particle density exceeds the fluid density by several orders of magnitude, all right hand terms can be neglected with respect to the viscous drag [57]. After some algebraic rearrangement the following equation results:

$$\frac{d\vec{U}_p}{dt} = \frac{\vec{U} - \vec{U}_p}{\tau}$$

(3.9)

where \(\tau\) is the velocity relaxation time of a particle:

$$\tau = \frac{4 \rho_p d_p^2}{3 c_d Re_d \mu}$$

(3.10)

In Tedeshi et al. [88], methods are given that allow to calculate the drag coefficient for spherical particles for a variety of flow conditions (relative Mach number, relative Reynolds number and Knudsen number).

In compressible flow a velocity change is accompanied by a temperature change. Therefore in addition to a velocity lag, a particle also incorporates a temperature lag. The time rate change of the particle temperature is given by [68]:

$$\frac{\pi d_p^3}{6 \rho_p c} \frac{dT_p}{dt} = h \frac{d_p^2}{(T - T_p)}$$

(3.11)

where \(c\) is the thermal heat capacity of the particle material and \(h\) is the heat transfer coefficient. This equation can be rewritten in order to obtain the temperature relaxation time in a similar way as the velocity relaxation time:

$$\frac{dT_p}{dt} = \frac{T - T_p}{\tau_T}$$

(3.12)

where the temperature relaxation time can be expressed as:

$$\tau_T = \tau \left[ \frac{1}{8} \frac{Pr}{Nu} \frac{c_d}{c_p} \right]$$

(3.13)
The Nusselt number is defined as \( Nu = \frac{h_d}{k} \), where \( k \) is the thermal conductivity of air.

To get an estimate of the temperature relaxation time the above equation is evaluated with the conservative assumption of pure conduction \( Nu = 2 \), Stokes’ drag \( c_d = 24/Re_d \), \( Pr = 0.71 \). The thermal heat capacity of TiO\(_2\) is \( c = 712 \) J/(kg K) which results in a temperature relaxation time of:

\[
\tau_T = 0.76\tau \quad (3.14)
\]

This shows that the particles will be in thermal equilibrium with the gas flow downstream of the slip region.

### 3.3.2 Determination of particle time response by means of a shock wave test

To establish the proper value of the particle time response under realistic flow conditions, it is evaluated experimentally by means of a shock wave test [88]. The trajectory of a particle over a shock wave is obtained by solving the ordinary differential equation (3.9) with the appropriate boundary conditions \( u_p(t = 0) = u_0 \) and \( u_p(t \to \infty) = u_1 \):

\[
u^* = \frac{u_p - u_1}{u_0 - u_1} = \exp \left( \frac{-t_p}{\tau} \right).
\]

Where \( u_p \) is the particle velocity, \( u_0 \) is the velocity upstream and \( u_1 \) is the velocity downstream of the shock, see figure 3.10.

![Figure 3.10: Particle response to a normal shock wave](image)

The above solution applies to the velocity component normal to the shock wave. This expression is integrated in time to obtain the particle velocity as a function of
particle position $x_p$ with initial condition $(x_0, t_0) = (0, 0)$:

$$x_p = u_1 \tau \ln \left( \frac{u_0 - u_1}{u_p - u_1} \right) + \tau (u_0 - u_p) \quad (3.16)$$

For shock waves having a low normal Mach numbers $M_{n,0} \leq 1.3$ and atmospheric total temperatures $T_t \approx 288 \, K$, the factor $(u_0 - u_p)$ may be neglected with respect to $u_1 \ln \left( \frac{u_0 - u_1}{u_p - u_1} \right)$ and an exponential function for the relaxation length $\xi$ may be defined:

$$u^* = \exp \left( \frac{-x_p}{\xi} \right) \quad (3.17)$$

At $t = \tau$, the particle is located at the position $x_p = \xi$ and $u^* = \exp(-1)$, the relaxation length can be related to the relaxation time using equation 3.16:

$$\xi = \tau |u_0 - (u_0 - u_1) \exp(-1)| \quad (3.18)$$

The above equation is similar to that obtained by Haermann et al. [35]. They however, define the relaxation length $\xi$ based on the criterion that $t = 2\tau$, so that $u^* = \exp(-2)$.

Using equation (3.17), the relaxation length can be directly obtained from the slope of the relaxation curve, measured by PIV, when displayed in a semi-logarithmic diagram:

$$\frac{d \ln (u^*)}{dx_p} = \frac{-1}{\xi} \quad (3.19)$$

The linearity of the log-plot enables to make a less ambiguous estimate of the relaxation length in comparison to the data fit procedures performed over linear scales [71], [44].

### 3.3.3 Assessment of the particle time response

The equations derived in section 3.3.2 allow the implementation of a convenient and unambiguous approach to determine the particle relaxation time and length from the spatial distribution of the velocity field. Several TiO$_2$ particle types were tested having different crystal sizes, see table 3.1. The first four types are manufactured by Kemira while the largest grade is a Dupont product. Although the specific weight of TiO$_2$ is 4, the bulk density reported by the manufacturers is considerably lower. The Kemira UV-TITAN products are reported to have a bulk density of 200 $kg/m^3$ while the other particle types range from 400 to 800$kg/m^3$. The crystal size reported in table 3.1 should not be interpreted as the particle diameter. Due to the small size, particle agglomeration occurs leading to an effective particle size about ten times larger [76]. Furthermore, the storage conditions (primarily humidity) as well as the distribution procedure affect the final particle (agglomerate) size to an important extent.

The shock wave test was performed in the TST27 wind tunnel [71] at Mach 2.1. The field of view was aligned with the shock such as to optimize the spatial resolution.
normal to the shock, see figure 3.11(a). A window size of 0.2 mm and a time separation of 600 ns was chosen in order to provide enough spatio-temporal resolution to sample the particle velocity decay across the shock wave (25 data points). For more information regarding the setup the reader is referred to [73].

In figure 3.12 the logarithmic plot is given in combination with the inferred relaxation length values. For small values of $u_*$ the velocity profiles in Figure 3.12 are affected by measurement errors (mainly optical distortions [30], [29]). Therefore only the data points for $u_*>0.1$ (corresponding to $0<x<2$ mm) are used to obtain the fit. Using the value of the relaxation length in combination with equation (3.18), the relaxation time is determined. In figure 3.13 the resulting fit described by equation 3.16 is plotted. The numerical values for $\tau$ and $\xi$ are summarized in table 3.1. From the numerical values and the curves may be concluded that in general the particle slip effect increases slightly with the size of the crystal diameter. However, the dependence is weak due to particle agglomeration. In particular, the 30 nm particles exhibit a larger relaxation time due to an increased level of agglomeration.
Particle slip compensation

Under steady flow conditions the particle acceleration can be derived from the particle velocity variation along its trajectory. In the present discussion a 1-dimensional formulation is used to introduce the concept:

$$\frac{d u_p}{d t} = \frac{\partial u_p}{\partial x_p} \frac{d x_p}{d t} = \frac{\partial u_p}{\partial x_p} u_p$$

(3.20)

The generalization to more dimensions reads as:

$$\frac{d \vec{U}_p}{d t} = \left( \nabla \vec{U}_p \right) \frac{d \vec{x}_p}{d t} = \left( \nabla \vec{U}_p \right) \vec{U}_p$$

(3.21)

Combining the above equation with equation 3.9 results in an expression for the flow velocity:

$$\vec{U} = \vec{U}_p + \tau \left[ \left( \nabla \vec{U}_p \right) \vec{U}_p \right]$$

(3.22)

Where the second term can be interpreted as a particle slip correction on the measured (particle) velocity. To test the potential of this technique and to assess the susceptibility to measurement noise, it is applied to the velocity fields used to obtain the particle relaxation time. A robust estimate of the first spatial derivative is provided by a local quadratic least-squares fit on a $7 \times 7$ kernel of the velocity field. The relaxation time used in the procedure is equal to that as obtained in section 3.3.3. In figure 3.14 the computed particle acceleration is shown normalized by the theoretical maximum acceleration:

$$a_{\text{max}} = -\frac{u_0 - u_1}{\tau}$$

(3.23)

Maximum acceleration levels between $0.4a_{\text{max}}$ and $0.5a_{\text{max}}$ are measured. The smoothing effect of the PIV interrogation algorithm and the kernel used to compute the spatial first derivative cause the maximum acceleration to be lower than that obtained from theory. The corrected velocity profiles are shown in Figure 3.15 in combination with the measured profiles. The increase in velocity gradients can be clearly observed as well as a small upstream shift of the shock wave origin. In Table 3.2 the effect of the particle slip filter is quantified. The distance is measured for a particle velocity decrease from $u^* = 0.9$ to $u^* = 0.15$, which is denoted by $x_{\text{slip}}$. A more convenient number would be $u^* = 0.1$ however in this velocity region the profile is affected by the optical blurring of the shock which makes this value less suitable. It is shown that the errors due to the particle slip can be decreased by approximately 50 % when applying
Figure 3.14: Acceleration computed for the measured velocity profiles

Figure 3.15: Measured (dotted line) and corrected (solid line) velocity profiles

<table>
<thead>
<tr>
<th>$d_c$</th>
<th>$x_{slip \ no\ correction \ [mm]}$</th>
<th>$x_{slip \ with \ correction \ [mm]}$</th>
<th>$\Delta x_{shock \ [mm]}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>30</td>
<td>1.52</td>
<td>0.72</td>
<td>0.33</td>
</tr>
<tr>
<td>50</td>
<td>1.10</td>
<td>0.54</td>
<td>0.31</td>
</tr>
<tr>
<td>170</td>
<td>1.12</td>
<td>0.56</td>
<td>0.31</td>
</tr>
<tr>
<td>240</td>
<td>1.18</td>
<td>0.61</td>
<td>0.33</td>
</tr>
<tr>
<td>550</td>
<td>1.66</td>
<td>0.92</td>
<td>0.33</td>
</tr>
</tbody>
</table>

Table 3.2: Particle slip estimate based on a velocity decrease from $u^* = 0.90$ to $u^* = 0.15$ and shock position shift based on 0.90$u^*$

Simultaneously the particle slip filter causes an upstream shock position shift of 0.3 mm due to the evaluation of the velocity derivatives.

### 3.4.1 Application to double ramp flow

The particle slip correction procedure is applied to the measured flow field over a 15°-45° double compression ramp at Mach 7.5 (see chapter 5). Due to the large second ramp angle an attached shock is not possible resulting in a strong curved shock. In figure 3.16 the flow field over the second compression ramp is visualized, where the strong curved shock is blurred due to particle slip. In the experiments TiO$_2$ particles were used having a crystal size of 50 nm (UV-TITAN L830) therefore a value of $\tau = 2.2 \ \mu s$ is used for the particle slip correction procedure. In Figure 3.17 the corrected velocity field is shown. It can be seen that the blurred region in the vicinity of the curved shock is visibly reduced. Away from the shock the correction does not yield significant changes to the flow field except for a slight increase in the noisy fluctuations since the correction term contains spatial derivatives.

Additionally in figure 3.18 the streamlines are shown both for the original and corrected velocity fields. When the corrected streamline crosses a shock it deflects earlier with respect to the original streamline. Since a streamline is an integral property, the correction procedure has a lasting effect on the trajectory. In regions where
particle slip does not occur also the streamline trajectory does not change. Finally in Figure 3.19 profiles are shown at \( y = 20 \, \text{mm} \) illustrating the original and corrected velocity and acceleration levels. At the location where the profile crosses the shock the velocity magnitude decreases strongly. At this location a maximum is found in the acceleration indicating a large particle slip whereas along the rest of the profile the acceleration levels are small. Comparing the original and the corrected velocity profiles confirms that the correction is the largest near to the shock and small elsewhere. In the experimental results a turbulent free shear layer is observed. The current procedure only uses the mean flow field for the correction. The extension of the procedure for unsteady high-speed flow conditions remains to be explored.

Figure 3.16: PIV result for the flow over the double compression ramp at Mach 7
Figure 3.17: Resulting velocity field after particle response correction

Figure 3.18: Effect of slip correction on the streamline pattern
Figure 3.19: Velocity profile at \( y = 20 \, \text{mm} \) with the corresponding computed acceleration level (green)
4.1 Introduction

In aerodynamics infrared thermography is generally applied for the study of boundary layer behaviour (notably transition and separation) as well as for quantitative heat flux measurements. In case of boundary layer investigations a qualitative description of the surface temperature is sufficient. However, for the calculation of the surface heat transfer, quantitative temperature information is required. Moreover a mathematical model has to be established that relates the surface temperature to the convective heat flux.

The determination of the surface temperature is performed by the measurement of infrared radiation emitted by the object surface within a given range of wavelengths. In order to accurately obtain the surface temperature from the radiation measurement, the measurement chain must be carefully modelled. In this chapter the basics of infrared theory will be presented. Thereafter, the infrared systems used in the present investigation are introduced. Finally the numerical data reduction techniques for the determination of the surface heat flux are discussed.

4.2 Infrared theory

4.2.1 Thermal radiation

Any object at a temperature above absolute zero radiates a certain amount of energy in the form of electromagnetic radiation. This thermal radiation is related to the temperature of its surface. The spectral distribution of the radiation emitted by a black body is described by Planck’s law [49]:

\[ E_{b,\lambda} = \frac{C_1}{\lambda^5 \exp \left( \frac{C_2}{\lambda T_s} \right) - 1} \]

(4.1)

where \( C_1 = 3.7415 \times 10^{-16} \, Wm^2 \) is the first radiant constant and \( C_2 = 1.4388 \times 10^{-2} \, mK \) is the second radiant constant. From equation (4.1) and figure 4.1 it can be
seen that the monochromatic radiation $E_{b,\lambda}$ varies with wavelength $\lambda$ and the surface temperature $T_s$. Planck’s law is valid for a black body (perfect radiator), which is a body that emits and absorbs at any temperature the maximum possible amount of radiation at any given wavelength. Following a curve in figure 4.1, the spectral emittance is zero at $\lambda = 0$ and increases to a maximum at a wavelength $\lambda_{\text{max}}$ then it decreases again until it approaches zero at large wavelengths. When the temperature increases, the wavelength for maximum monochromatic emissive power decreases. The relationship between $\lambda_{\text{max}}$ and the absolute surface temperature $T_s$ is called Wien’s displacement law

$$\lambda_{\text{max}} T_s = 2.898 \times 10^{-3} \ [m \cdot K]. \quad (4.2)$$

This formula expresses the common observation that for increasing radiator temperature its color varies from red to orange to yellow, the color corresponding with $\lambda_{\text{max}}$. When Planck’s law is integrated over all wavelengths, the total emitted power per unit surface area is given by the Stefan-Boltzmann equation:

$$E_b = \int_0^\infty E_{b,\lambda} \, d\lambda = \frac{qr}{A} = \sigma T_s^4, \quad (4.3)$$

where $A$ is the surface area, $\sigma = 5.670 \times 10^{-8} \ W/(m^2 \ K^4)$ is the Stefan-Boltzmann constant and $T_s$ is the surface temperature. $E_b$ represents the total area under each curve in figure 4.1 for a given temperature.

### 4.2.2 Real emitters

Planck’s law is valid for black body radiators, however real objects never behave as black bodies, although they may approach black body behavior in certain spectral
intervals. A real body can interact with incident radiation $I$ in three ways; a fraction of the incident radiation $I_\alpha$ may be absorbed, a fraction $I_\rho$ may be reflected and a fraction $I_\tau$ may be transmitted. Since all these fractions are wavelength dependent, the subscript $\lambda$ should be used. The sum of these coefficients adds up to unity at any wavelength as a result of energy conservation:

$$\alpha_\lambda + \rho_\lambda + \tau_\lambda = 1$$  \hspace{1cm} (4.4)

The amount of radiation emitted by a surface at a certain temperature can be correlated to the radiation emitted by a black body at the same temperature by the spectral emissivity coefficient $\epsilon_\lambda$ defined as:

$$\epsilon_\lambda = \frac{E_{o,\lambda}}{E_{b,\lambda}}$$  \hspace{1cm} (4.5)

where $E_{o,\lambda}$ is the monochromatic radiation of a real surface. In order to remove the wavelength dependence the gray body hypothesis is introduced for which the emissivity is a constant at each wavelength $\epsilon_\lambda = \epsilon < 1$, see figure 4.2:

$$\epsilon = \int \epsilon_\lambda \frac{E_{b,\lambda}}{E_b} d\lambda$$  \hspace{1cm} (4.6)

Finally Kirchhoff’s law states that the spectral emissivity is equal to the spectral absorptivity for any specified temperature and wavelength, $\epsilon_\lambda = \alpha_\lambda$.

Since a gray body has a constant spectral emissivity, its total emitted radiation can be related to the total radiation emitted by a black body using the Stefan-Boltzmann equation (4.3)

$$E_o = \epsilon \sigma T^4_s.$$  \hspace{1cm} (4.7)

Figure 4.2: Gray body radiator [49]

Figure 4.3: Variation of directional emissivity for several electrical nonconductors [49]
Using this equation and Kirchoff’s law $(\epsilon = \alpha)$ the net radiant energy transfer per unit surface area of the surroundings to or from a gray body at a certain temperature can be calculated. The radiated power by the gray body is

$$E_r = E_o - E_a = \varepsilon_o \sigma (T_o^4 - T_a^4).$$

(4.8)

where $E_a$ is the amount of radiant energy received from the surroundings and $E_o$ is the amount of energy emitted by the gray body. For real surfaces the emissivity is also a function of the temperature and the observation angle $\theta$. When applying the gray-body hypothesis, the average emissivity is used for the wavelength band where the bulk of radiation is emitted. Since this wavelength band changes with temperature also the gray-body emissivity will change with surface temperature.

The variation of emissivity with the optical angle $\theta$ may be large. A polar plot illustrating the directional emissivity for some electrical non-conductors is shown in figure 4.3. In these graphs $\theta$ is the angle between the normal and the direction of the radiant beam emitted from the surface. If the emissivity is independent of the optical angle, the emissivity curves would be semi-circles. Figure 4.3 illustrates that for nonconductors such as wood, paper and oxide films the emissivity decreases at large values of the emission angle $\theta$. For practical purposes the directional emissivity may be regarded as constant for $\theta < 50^\circ$.

### 4.2.3 Environmental disturbances

Radiant energy also interacts with fluid media, it gets absorbed, reflected and emitted energy depending on the fluid properties. Unless special precautions are taken when measuring the radiation emitted by an object (e.g. vacuum chamber) the surrounding air will reflect and absorb radiation, disturbing the measurement. The magnitude of the disturbance is influenced by a large number of factors, for example: air temperature, air pressure, relative humidity, amount of particles suspended in the air, distance covered by the radiation through the medium. Most of these parameters can be neglected at standard conditions and the reflectivity may be assumed zero. The radiant budget (4.4) now becomes: $\alpha_{air} + \tau_{air} = 1$. The two parameters that strongly affect the transmissivity are the distance covered by the radiation and the amount of chemical components $H_2O$ and $CO_2$. Water vapor and carbon dioxide absorb energy at certain wavelengths in the infrared spectrum, leading to a decrease in transmissivity. Wavelength areas of reduced transmissivity can be seen in figure 4.4.

The region up to the wavelength $\approx 5 \mu m$ is mainly influenced by the absorbing affect of $CO_2$. Air is nearly opaque between 5 and 8 $\mu m$, this is caused by the water vapor in the air. Wavelength regions of high transmissivity are called atmospheric windows. Especially for radiation measurements over long distances (> 10 m) it is important to operate within these atmospheric windows. For this reason most infrared detectors operate in two distinct bands:

- Short Wave Band (SWB) 3 $\mu m$ – 5 $\mu m$,
- Long Wave Band (LWB) 8 $\mu m$ – 12 $\mu m$.

The choice of the band depends on the expected temperature range, sensitivity, detectivity, simplicity of the infrared detector etc. When applying Wien’s displacement
law (equation (4.2) and figure 4.1), the maximum emissive power of objects at ambient temperature falls into the long wave band, for example a body with a temperature of 293 $K$ emits maximum power at $\lambda_{\text{max}} = 9.9 \, \mu m$.

## 4.3 Infrared measurement systems

Basically there are two approaches to the detection of infrared radiation. First is the single sensor approach which is used by older high-end systems up to the late 1990’s. Here the incident radiation is integrated over the instantaneous field of view (IFOV) and the measured signal is a function of time only. By using a scanning system to displace the IFOV of the single sensor, a system is obtained that scans the desired field of view (FOV). The second approach uses an array sensor, where the incident radiation is integrated in time and the response is a function of the spatial variables. In the current state-of-the art systems a CCD is used for the infrared signal detection.

Infrared radiation can be measured using two types of detectors. First there are thermal detectors, which transform the incident luminous flux into heat by absorption. The resulting temperature variation produces the output signal. Examples of thermal detectors are bolometers (conductivity change due to temperature increase), pyroelectric detectors (production of surface electric charges due to heating), thermopiles (production of a voltage due to temperature increase) and pneumatic detectors (gas expansion due to heating). Secondly there are quantum detectors, which are
superior to the thermal detectors in terms of sensitivity. This type measures the direct excitation of its electrons to conduction states by incident photons (photoelectric detectors). In the investigation two infrared systems (Agema 880 LWB and CEDIP Titanium 530L) were used that are both equipped with quantum detectors.

### 4.3.1 Agema 880 LWB system

The system consists of an Agema Thermovision 880 LWB scanner and a BRUT (Burst Recording Unit) system for image storage and analysis. The detector is a Mercury Cadmium Telluride (MCT) quantum detector [67]. The sensor has a spectral response of 8 – 12 $\mu m$ and is mounted at the bottom of a $LN_2$ (liquid nitrogen) filled Dewar chamber which cools the sensor to a temperature of 77 K. The cooling has two beneficial effects, first the cooling will induce a large temperature difference between the object and the sensor which results in a larger net radiant energy transport to the sensor (see equation (4.8)) while secondly, the low temperature raises the signal to noise ratio of the sensor. The camera has a single quantum detector with a scanning mechanism. This provides the displacement of the IFOV to build up a complete frame, which consists of $140 \times 280$ pixels at an interlace mode of 4:1. Due to the mechanical scanning mechanism it has a relatively low frame frequency of 6.25 Hz. The camera can operate at higher imaging speeds when it is switched into line scan mode by stopping the oscillating mirror responsible for the vertical displacement of the IFOV. This results in an imaging frequency of 2.5 kHz. Although a significant speed increase is achieved in this way it comes at the expense of measuring over a line only, therefore losing information on the second spatial direction. The BRUT system is used to process and transfer the acquired data. The analogue video signal is first processed in the analogue section then converted to digital data by a 13 bit AD converter. The completed frames are transferred to the sequence memory (RAMdisk) for later playback or storage. Since the mirrors of the scanning system are driven by electronic motors, electrical fields are built up which contribute to the noise in the signal of the sensor. An appropriate parameter to describe the influence of noise is the "Noise Equivalent Temperature Difference" (NETD). In this quantity the total sensor noise is expressed as an equivalent temperature difference. For the Agema 880 scanner the NETD is $\pm 200 \ mK$.

---

**Table 4.1: System characteristics of the IR systems**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Agema 880 LWB</th>
<th>CEDIP Titanium 530L</th>
</tr>
</thead>
<tbody>
<tr>
<td>Detector type</td>
<td>Mercury Cadmium Telluride</td>
<td>Mercury Cadmium Telluride</td>
</tr>
<tr>
<td>Sensor</td>
<td>Scanning mechanism</td>
<td>Focal plane array</td>
</tr>
<tr>
<td>Spectral response [$\mu m$]</td>
<td>8-12</td>
<td>7.7-9.3</td>
</tr>
<tr>
<td>NETD [mK]</td>
<td>200</td>
<td>25</td>
</tr>
<tr>
<td>Full frame rate [Hz]</td>
<td>6.25</td>
<td>250</td>
</tr>
<tr>
<td>Max frame rate [kHz]</td>
<td>2.5 @ 140 × 1 pix</td>
<td>20 @ 64 × 2 pix</td>
</tr>
</tbody>
</table>
The CEDIP Titanium 530L system

The CEDIP Titanium 530L is a more recent system that consists of an infrared camera and a personal computer to store and visualize the data. The detector is a focal plane array of MCT sensor material. The detector has 320 × 256 pixels, a spectral response of 7.7 – 9.3 µm and a maximum frame rate of 250 Hz at full resolution. The sensor is cooled by an integral Stirling cycle and has a NETD of 25 mK. The sensitivity of the camera can be optimized for a given temperature range by changing the integration time. In practice an integration time of 340 µs is used for a temperature range of $-20 \, ^\circ C < T < 50 \, ^\circ C$ and 72 µs for $50 \, ^\circ C < T < 200 \, ^\circ C$. In order to correct for non-uniform pixel behavior, a one-point or two-point non-uniformity correction (NUC) is performed before utilizing the camera. The one-point NUC corrects for the pixel signal offset, while the two-point NUC also corrects for the pixel gain.

4.3.3 Calibration procedure

Both IR systems are calibrated using a reference black body simulator. The camera is calibrated by correlating the temperature of the black body simulator to the output of the camera. The black body simulator is heated to a prescribed temperature which is measured by means of a thermocouple, simultaneously the raw camera output is collected. This process is repeated for different temperatures the temperature range that is expected during the experiments. For the Agema system a camera function is provided by the manufacturer that is used to obtain a curve fit to the calibration points. In case of the CEDIP system, the calibration points are linearly interpolated to obtain the surface temperature from the camera output.

The black body simulator used for the calibration should have an emissivity close to 1. This is achieved by constructing a cavity which minimizes the reflection of incident radiation. Although the actual internal emissivity is typically 0.9 when painted, the effective emissivity of the complete system may approach 0.999.
The black body simulator consists of a circular cavity, see figure 4.7 for a schematic. The inside of the cavity is painted black (\( \epsilon \approx 0.9 \)) to increase the emissivity of the walls. To maximize the amount of internal reflections of incident radiation and thus the amount of radiation that is absorbed, a cone is placed at the back of the cavity and the opening is narrowed.

The black body is heated uniformly by circulating hot water through a hollow copper coil which is wound around the outside of the cylinder. Two channels are drilled in the cone which are connected to the coil for faster heating of the cone. The outside of the black body is insulated with glasswool to minimize heat losses therefore further improving the uniformity of the temperature field inside the cavity. When the black body is heated, following Kirchoff’s law, a source of radiation is created since a perfect absorber is also a perfect emitter. In Mayer [56] the apparent emissivity was calculated for different values of the relative aperture \( R_i/R \) and the length-diameter ratio \( L/R \). If \( R_i/R \) decreases then the apparent emissivity increases, the minimum size of the opening is determined by the camera resolution and the need to average over a certain amount of pixels to decrease the influence of camera noise. As expected, the emissivity also increases with increasing \( L/R \). The black body used in the calibration has a length \( L = 200 \text{ mm} \), a cylinder diameter \( R = 25 \text{ mm} \) and an opening with a diameter \( R_i = 10 \text{ mm} \). For this geometry, according to Mayer [56], an apparent emissivity of 0.999 is achieved for an internal emissivity of 0.75. For the current black body, the emissivity of the walls is approximately 0.9, which increases the apparent emissivity even more. Since a germanium window is used during the wind tunnel experiments this is also included in the calibration setup [25]. The camera monitors the radiation emitted by the black body through the germanium window, see figure 4.8. In this way the reduced transmissivity due to the germanium is incorporated into the calibration. The window is put at an angle because during the experiments the camera also looks at the model at an angle in order to avoid the reflection of the camera to occur in the field of view (narcissus effect). The temperature of the black

![Figure 4.7: Black body layout](image)

![Figure 4.8: Setup for the Calibration measurements](image)
body is measured by a thermocouple attached to the back of the cone since this is the part directly monitored by the camera.

### 4.3.4 Environmental disturbance correction

For an accurate surface temperature measurement, the measured camera value has to be corrected for the radiation emitted by surrounding environment. When the camera views the wind tunnel model, it also registers radiation from the ambient that is reflected from the surface of the model. The radiation received by the camera consists of two contributions (the transmissivity is assumed to be zero so that \( \rho = 1 - \epsilon \)):

\[
I_{\text{meas}} = \epsilon \cdot I_o + (1 - \epsilon) \cdot I_a.
\]

Here \( I_{\text{meas}} \) is the intensity that is directly measured by the infrared camera, whereas \( I_o \) is the intensity due to the radiation of the model. The radiation emitted by the environment \( I_a \) is obtained from the ambient temperature and \( \epsilon \) is the emissivity of the wind tunnel model. If the equation is rewritten, \( I_o \) can be corrected according to:

\[
I_o = I_{\text{meas}} - (1 - \epsilon) \cdot I_a / \epsilon.
\]  \hspace{1cm} (4.9)

Finally the surface temperature is evaluated using a calibration table or a fitting function.

### 4.4 Heat flux data reduction

#### 4.4.1 Heat flux sensors

Heat-flux sensors generally consist of slabs with a known thermal behavior, which temperature is measured at fixed points. The measured temperature is correlated to the convective heat transfer rate using the equation for heat conduction in solids in combination with a proper sensor model.

Commonly one-dimensional heat-flux sensors are used, where the deduced heat-flux is assumed to be (predominantly) normal to the sensing-element surface. The temperature gradient in the direction parallel to the plane of the slab is neglected with respect to that normal to the sensor surface, which requires the usage of a low conductive material. In practice, the slab surfaces may also be curved, however the curvature effects are neglected if the thickness of the layer affected by the input heat-flux is small compared to the local radius of curvature of the slab.

In the following, only ideal one-dimensional sensors are considered. The term ideal means that the thermophysical properties of the sensor materials are assumed independent of the temperature (which is true if the temperature range is not very large) and that the effect of the presence of the actual temperature-sensing element is not considered, this is valid for IR thermography where the paint on the surface or the model surface itself is the sensor.
There are four basic one-dimensional sensor models [16], the thin-film sensor, thick-film sensor, wall calorimeter or thin-skin method and gradient sensor. The heat transfer gauges most commonly used in short duration experiments are of the thin film type [78]. In the classic implementation a very thin resistance thermometer measures the surface temperature of a “thermally” thicker slab on which it is mounted. The heat flux is inferred from the theory of heat conduction in a semi-infinite one dimensional wall. Other methods of measuring the surface temperature are: surface-mounted thin film thermocouples, temperature sensitive paint, liquid crystals or infrared thermography. When observing the surface temperature using an infrared camera the actual gauge is the coating (paint) used to obtain a high emissivity or the model surface itself. Although the temperature measuring methods may differ significantly from each other, the principles for heat flux data reduction will be the same to a large extent.

4.4.2 One dimensional semi-infinite model

The heat transfer data reduction technique for the thin film approach is based on the one dimensional semi-infinite model. In this case the heat flux can be expressed as a function of the surface temperature variation in time. The heat conduction into a semi-infinite one-dimensional wall is modeled by the Fourier equation:

\[
\frac{\partial T(y, t)}{\partial t} = \frac{k}{\rho c} \frac{\partial^2 T(y, t)}{\partial y^2} \tag{4.10}
\]

with the appropriate boundary conditions:

\[-k \frac{\partial T(0, t)}{\partial y} = q_s(t) \quad \text{at} \quad y = 0, \]

\[T(y, 0) = T_i \quad \text{at} \quad t = 0, \tag{4.11}\]

\[T(\infty, t) = T_i \quad \text{at} \quad y = \infty.\]

where \(\rho\) is the density, \(k\) is the conductivity and \(c\) is the heat capacity of the wall material. In general, the grouping \(\rho ck\) is referred to as the thermal product and \(\frac{k}{\rho c}\) as the thermal diffusivity.

![Figure 4.9: Semi-infinite one dimensional heat conduction](image)

In general, the solution of equation (4.10) gives the temperature as a function of the location \(y\), time \(t\) and the surface heat flux \(q_s(t)\), which is also a function of time.
However, in the current application the surface temperature is measured, which in fact is the solution at $y = 0$ and one of the boundary conditions, here $q_s(t)$, must be retrieved. This is called an inverse problem and in this case the general solution is [78]:

$$ q_s = \sqrt{\frac{\rho c k}{\pi}} \int_0^t \frac{dT_s(\tau)}{\sqrt{t-\tau}} d\tau. \quad (4.12) $$

This equation describes the surface heat transfer in time as a function of the surface temperature $T_s(t) = T(0, t)$. However this form is inconvenient for practical data reduction since it involves the differential of the surface temperature which will make the heat flux inaccurate due to the presence of noise in the measured temperature signal. The differential can be removed by integrating equation (4.12) by parts with the boundary condition, $T_s(\tau) = 0$ for $\tau = 0$ [18]:

$$ q_s = \sqrt{\frac{\rho c k}{\pi}} \left[ \frac{T_s(t) - T_{s,i}}{\sqrt{t}} + \frac{1}{2} \int_0^t \frac{T_s(t) - T_s(\tau)}{(t-\tau)^{3/2}} d\tau \right]. \quad (4.13) $$

where $T_{s,i}$ is the initial surface temperature. This is a more convenient form of the solution for data analysis at non constant heat transfer rate. Due to a singularity in the integral term at $t = \tau$ errors will be introduced in the deducted values of $q_s$. Numerical techniques which either reduce or avoid this difficulty are discussed in section 4.4.3.

For a constant heat transfer $q_s(t) = \text{const}$, equation (4.12) reduces to:

$$ T_s - T_{s,i} = \frac{2q_s}{\sqrt{\pi}} \sqrt{\frac{t}{\rho c k}}. \quad (4.14) $$

This shows that for a constant heat transfer the surface temperature develops as a square root in time. It also shows that for a given value of the surface heat flux $q_s$, the temperature signal $T_s - T_{s,i}$ is maximized by selecting a material having a low thermal product.

For convective heat transfer, the surface heat flux may be expressed as a function of the temperature difference between the wall temperature $T_s$ and the adiabatic wall temperature and the convective heat transfer coefficient $h$ as:

$$ q_s(t) = h (T_{aw} - T_s(t)) \quad (4.15) $$

Now the one dimensional heat diffusion equation in solids is evaluated using the following surface boundary condition:

$$ -k \frac{\partial T}{\partial y} |_{y=0} = h [T_{aw} - T(0, t)], \quad (4.16) $$

Since $h$ is constant, an analytical expression may be formulated for the normalized temperature development with time [49]:

$$ \frac{T(y, t) - T_{s,i}}{T_{aw} - T_{s,i}} = \text{erfc} \left( \frac{y}{2\sqrt{\alpha t}} \right) - \exp \left[ \left( \frac{hy}{k} + \frac{h^2yt}{k^2} \right) \right] \text{erfc} \left( \frac{y}{2\sqrt{\alpha t}} + \frac{h\sqrt{\alpha t}}{k} \right), \quad (4.17) $$
This equation may be used to determine the development of the surface temperature by putting \( y = 0 \) in equation (4.17):

\[
T_s = T_{s,i} + (T_{aw} - T_{s,i}) \left[ 1 - e^{\beta^2} \operatorname{erfc}(\beta) \right],
\]

(4.18)

where the introduction of the non-dimensional parameter \( \beta = h\sqrt{t}/\sqrt{\rho c k} \) allows to simplify the notation.

### 4.4.3 Data reduction techniques

The evaluation of the heat transfer rate from the observed surface temperature time history record can be carried out by means of numerical integration or by means of curve fitting techniques.

For numerical integration, equation (4.13) may be used:

\[
q_s = \frac{\sqrt{\rho c k}}{\sqrt{\pi}} \left[ \frac{\phi(t)}{\sqrt{t}} + \int_0^t \frac{\phi(t) - \phi(\tau)}{(t - \tau)^{3/2}} d\tau \right],
\]

(4.19)

where \( \phi(t) \) is the variation in the surface temperature relative to the initial temperature, \( T(t) - T(0) \). Numerical integration of the second part of the expression enables to determine the non-constant part of the heat transfer. To prevent the singularity at \( t = \tau \), proper numerical procedures (e.g. trapezoidal rule for integral evaluation) can be applied. The error introduced by the singularity is largest for small values of \( t \) while maintaining constant steps of integration. In [18] Cook and Felderman developed a direct numerical integration method by approximating \( \phi(\tau) \) through a piecewise linear function of the form:

\[
\tilde{\phi}(\tau) = \phi(t_{i-1}) + \frac{\phi(t_i) - \phi(t_{i-1})}{\Delta t} (\tau - t_{i-1})
\]

(4.20)

where \( t_{i-1} \leq \tau \leq t_i \) and \( i = 1, 2, 3, \ldots, n \). This equation can now be inserted into equation (4.19), which after some algebra (see [78] for details) results in:

\[
q_s(t_n) = \frac{\sqrt{\rho c k}}{\sqrt{\pi}} \left[ \frac{\phi(t)}{\sqrt{t}} + \sum_{i=1}^{n-1} \left\{ \frac{\phi(t_n) - \phi(t_i)}{(t_n - t_i)^{1/2}} - \frac{\phi(t_n) - \phi(t_{i-1})}{(t_n - t_{i-1})^{1/2}} \right\} + \frac{\phi(t_i) - \phi(t_1)}{(t_n - t_i)^{1/2} + (t_n - t_{i-1})^{1/2}} \right] + \frac{\phi(t_n) - \phi(t_{n-1})}{\sqrt{\Delta t}}.
\]

(4.21)

This equation can be directly applied to a measured temperature signal to calculate the heat transfer. The only approximation involved is the local linearization of \( \phi(t) \). As has already been noted, the time derivative of the measured temperature in the integrand of equation (4.12) will yield the amplification of any noise present in the measured temperature signal. So, also here integration by parts is used to avoid this noise amplification. An alternative approach to minimize the need for smoothing of the temperature signal is presented in Simeonides [81]. In effect, the one-dimensional heat conduction equation is solved with the same assumptions, initial and boundary conditions as before but now considering the cumulative heat input to the model.
surface during a time $t$ rather than the instantaneous convective heating rate. The result is:

$$Q(t) = \int_0^t q_s(\tau) d\tau = \sqrt{\frac{\rho c k}{\pi}} \int_0^t \frac{T_s(\tau)}{(t - \tau)^{1/2}} d\tau,$$

Equation (4.22) can be numerically integrated [1]:

$$Q(t_n) = \sqrt{\frac{\rho c k}{\pi}} \sum_{i=1}^{n} \frac{\phi(t_{i-1}) + \phi(t_i)}{(t_n - t_{i-1})^{1/2} + (t_n - t_i)^{1/2}} \Delta t.$$

Now the heat transfer rate can be computed using finite differencing:

$$q_s(t_n) = \frac{dQ(t_n)}{dt_n} = \frac{-2Q(t_{n-8}) - Q(t_{n-4}) + Q(t_{n+4}) + 2Q(t_{n-8})}{40(t_n - t_{n-1})}$$

Alternatively to the numerical integration technique also an algebraic curve fitting approach may be used [22], [75]. In this approach an algebraic solution is fitted to the measured surface temperature variation in time. The most simple one is the solution for a constant heat transfer, equation (4.14), where the heat transfer is obtained directly. Alternatively the solution for a constant heat transfer coefficient can be used, in this case the heat transfer coefficient and the adiabatic wall temperature are obtained. The benefit of this approach is that the technique is insensitive to noise and only few measurement points are necessary. The algebraic curve fitting techniques are not used in the current investigation since the numerical integration techniques proved to be sufficiently accurate and more flexible.

### 4.4.4 Error sources

In the development of the data reduction methods some assumptions have been made such as constant material properties and no lateral conduction.

Since the measurements are performed within a relative small temperature range 293 – 343 K, the material properties are not expected to change much. For example in [46] the thermal conductivity is given as a function of temperature for polycarbonate material with a nominal conductivity of $k = 0.23 \text{ W/(m K)}$. For a pressure of 1 bar and a temperature range between 393 K and 523 K an almost linear conductivity variation with temperature was found with slope: $\frac{dk}{dT} = 2 \times 10^{-4} \text{ W/(m K)}$. In the experiments therefore a maximum variation in $k$ of 5 % is expected. In [3], values are given for the variation of the specific heat capacity of polycarbonate with temperature. Within the range 320 – 380 K a $\frac{dc}{dT} = 3.3 \text{ J/(kg K)}$ was obtained. This gives a maximum variation of 14% in $c$ for a temperature range between 293 – 343 K. The temperature increase of 50 K taken for the assessment of the material properties is rather conservative, in practice the average temperature increase during a test is 20 K and the overall error is estimated to be in the order of 10%.

The contribution of lateral conduction is assessed using the maximum measured temperature difference over two neighbouring pixels. During the tests this temperature difference is 4 K with a pixel pitch of 0.55 mm. This results in a local lateral conduction of:

$$q_{\text{cond}} = k \frac{\Delta T}{\Delta x} = 1454 \text{ W/m}^2$$

(4.25)
In that same region the convective heat flux was 140 kW/m². So, compared to the convective heat flux (1%) the lateral conduction term can therefore be neglected.

### 4.4.5 Heat flux normalization

The surface heat flux is obtained from the temperature time variation by means of the indirect numerical integration technique of section 4.4.3. In boundary layer theory the normalization of the surface heat flux is commonly performed following the definition of the Stanton number:

\[
St = \frac{q_s}{\rho_e u_e (h_{aw} - h_w)}
\]  

(4.26)

\(\rho_e\) and \(u_e\) are the density and velocity at the boundary layer edge respectively, \(h_{aw}\) and \(h_w\) are the adiabatic wall enthalpy and wall enthalpy respectively. Physically the Stanton number relates the convective surface heat transfer to the convected enthalpy in the free stream. In case of a calorically perfect gas where \(h = c_p T\), it is defined as:

\[
St = \frac{q_s}{c_p \rho_e u_e (T_{aw} - T_w)}
\]  

(4.27)

In the current study the definition adopted for the Stanton number is somewhat different, to indicate this the alternative notation \(c_h\) is used instead of \(St\):

\[
c_h = \frac{q_s}{c_p \rho_\infty u_\infty (T_t - T_w)}.
\]  

(4.28)

Where the reference values for \(\rho\) and \(u\) are taken as the free stream values and the total temperature rather than the adiabatic wall temperature is used. The free stream values are used since \(\rho\) and \(u\) are not available at the boundary layer edge. In a short duration facility, the adiabatic wall temperature cannot be measured accurately therefore the total temperature is chosen as a reference using the following consideration: the classical definition for the adiabatic wall temperature is as follows [82]:

\[
T_{aw} = T_\infty + r \frac{u_\infty^2}{2c_p} = T_\infty \left\{ 1 + r \frac{\gamma - 1}{2} M_\infty^2 \right\}.
\]  

(4.29)

where \(r\) is the recovery factor. For high Mach numbers this equation may be approximated by:

\[
T_{aw} \approx r \frac{u_\infty^2}{2c_p} \approx r T_t.
\]  

(4.30)

Due to the uncertainty in the recovery factor for general flow situations the modified Stanton number, equation (4.28) is generally applied.
Double compression ramp flows

5.1 Introduction

As already was highlighted in the introductory chapter, the occurrence of shock wave boundary layer interaction is of paramount importance in the design of re-entry and sustained hypersonic flight vehicles. The interaction may occur in several locations on the vehicle as is shown in figure 5.1. The local geometry largely defines the type of interaction that occurs, in the current investigation two configurations are studied. The first features a double compression ramp where both ramp angles allow for attached shocks resulting in a relatively simple flow field. This flow resembles the interaction as it occurs on control surfaces and engine inlets. The second configuration uses a larger second ramp angle for which an attached shock is no longer possible causing a complicated interaction. This case has been selected in order to investigate the origin of the higher heat loads encountered in the reattachment region and the presumably high turbulent activity in the shear layers generated by shock interactions. This geometry represents the interaction as they occur at fuselage-wing intersections, ablated nose cones (see figure 5.2) and control surfaces in off-design conditions.

In this chapter the measurements of the flow over two compression ramp models will be discussed. In the experiments PIV, infrared thermography, schlieren and oil flow visualisation is used to investigate the flow. The observations from the measurement techniques will be discussed to form a complete description of the flow. Also a comparison will be made with respect to computational fluid dynamic results. The flow field over a double compression ramp is governed by inviscid effects away from the surface and viscous effects close to the wall and in the separated region. The outer flow field is dictated by shock interaction induced by the first and second ramp. For a given Mach number, the type of interaction depends on the deflection angles. In turn the nature of the interaction has a large impact on the reattachment process, since it may induce high pressure and temperature loads. Edney [27] identified six types of shock interference, his classification terminology has been universally adopted and will also be used in the following discussion. From the six types, interac-
Figure 5.1: Shock/shock and shock/boundary layer interactions [23]

Figure 5.2: Shock interaction pattern over a rough 'ideal' ablated nose shape [38]

Figure 5.3: Edney type shock interactions for double compression ramps, after [61]
In the inviscid representation of the shock interaction phenomenon as given above an oblique shock would emerge from the compression corner to attain the appropriate deflection of the flow. However there is a boundary layer present near the wall. Due to the no-slip condition at the wall also a subsonic sublayer is present. Through this subsonic sublayer information can be transmitted upstream and the incoming flow feels the effect of the deflected ramp in advance. With a moderate compression corner angle, a weak shock wave may extend into the boundary layer (the shock strength and inclination changes due to the Mach number variation along the wall-normal direction) but it must vanish at the interface between the supersonic part of the flow and the subsonic layer. Due to the upstream compression corner the subsonic part of the boundary layer will encounter a continuous pressure rise, which is bound to cause the streamlines to diverge and thickens the subsonic layer. This divergence will locally influence the outer hypersonic/supersonic stream causing the formation of a system of compression waves which will eventually coalesce onto a single oblique shock, see figure 5.4. For a modest compression corner angle, the resulting weak interaction will be contained within a relatively small region near the compression corner. As the compression corner angle is increased, the oncoming boundary layer faces an increasing adverse pressure gradient, until it can no longer withstand it and separates. In this case, the upstream influence of the compression corner is increased and the shock structure becomes significantly distinct from the inviscid case. The recirculation bubble which is formed in well separated cases is bounded by a separation shock upstream of the compression corner and a reattachment fan which is created when the shear layer approaches the reattachment point and is gradually deflected (through a series of compression waves) to turn parallel to the second ramp surface. The compression fan will gradually converge into a reattachment shock. Both the separation and reattachment shock will merge at some distance from the second ramp into a single stronger shock which is equal to the inviscid wedge shock. Downstream of reattachment due to re-compression the boundary layer will reach a minimum thickness in the “neck” region, where peak heating occurs.

![Viscid flow field over a separated compression corner](image-url)

Figure 5.4: Viscid flow field over a separated compression corner

The current chapter focusses on the description of the flow features over planar double compression ramps. Furthermore, by combining different measurement tech-
niques and CFD computations, the use of PIV and QIRT in the HTFD is validated under real experimental conditions. Two configurations are studied; a 15°-30° and 15°-45° double ramp. First the 2D flow field in the center of the model is studied. This is complemented by an experimental investigation of the spanwise variation of the flow over the model. Due to the finite span of the model, edge effects will influence the flow field, causing pressure differences in spanwise direction inducing flow spillage. The current investigation will assess the influence of edge effect on the overall flow structure. Furthermore secondary vortical structures are known to occur in proximity of the separated region. They will manifest themselves in the reattachment region where large heat loads are induced. These features are also addressed.

5.2 Wind tunnel model

In the investigation a planar wind tunnel model is used. This is preferred over an axisymmetric geometry since it maximizes the size of the flow phenomena and thus the measurability by means of PIV. Two model geometries were realized using a modular approach. The second ramp angle is varied by fitting an alternative second ramp module. The total model length is 150 mm and the width is 110 mm. The first part of the model has a length of 96 mm and features a 15° ramp with respect to the free stream. The second ramp modules have angles of 30° and 45° with respect to the free stream, they have a length of 66 mm and 81 mm respectively, see figure 5.5. The second ramp angles are chosen such as to generate an Edney type VI and V respectively.

For the application of infrared thermography, the model parts are produced in Makrolon, see section 2.3.2. For the PIV experiments the model is painted with an opaque black paint as primer and Rhodamine paint on top to minimize laser light reflections.

![Wind Tunnel Model Diagram](image)

Figure 5.5: Wind tunnel model configurations and dimensions

5.3 15-30 degree ramp flow

From schlieren observations (see below) it is confirmed that the Mach 7.5 flow over the 15°-30° double ramp features an Edney type VI interaction. The schematic flow
field obtained using the shock polar diagram is given in figure 5.6, where the separated region is modeled as an extra deflection of $4^\circ$.

![Figure 5.6: Schematic of the 15°-30° wedge shock pattern (left) and shock polar diagram (right)](image)

The schlieren image is given in figure 5.7; the leading edge shock, separation shock and reattachment shock are visualized. The three shocks appear to focus approximately towards a single point where the Type VI interaction occurs. From the intersection point a combined shock and an expansion wave separated by a slip-layer emerges. The single interaction point is specific for the current situation, in general two interaction points will be present in the flow. Normally, an interaction will take place between the leading edge shock and the separation shock and further downstream the reattachment shock interacts with the resulting shock from the first interaction.

The shock wave angles obtained from oblique shock wave theory, schlieren and PIV are respectively 21.6°, 21° and 22° for the leading edge shock and respectively 43.0°, 42° and 43° for the combined shock over the second ramp. From the values it can be concluded that they are in good agreement. Leading edge and reattachment shocks are clearly visualized by the PIV measurement (Figure 5.7). Also the slip-layer emanating from the shock intersection point is visible as a sharp velocity gradient in the direction normal to the wall. The slip layer discriminates the higher velocity (low entropy loss) flow passing through multiple shocks and the lower velocity (higher entropy loss) flow passing through a single stronger shock. The conjectured expansion wave is not sufficiently resolved with the PIV measurement, this is also the case for the flow properties close to the model surface due to lack of spatial resolution and laser light reflections from the wall.

On the first ramp the heat transfer profile gradually decreases downstream due to the spatial development of the boundary layer. When the boundary layer separates the heat transfer decreases, after which it increases strongly again as a result of flow compression on the second compression ramp. The maximum is attained at flow reattachment; downstream the heat transfer decreases again due to boundary layer development. The reattachment point inferred from schlieren visualization coincides with the heat transfer peak measured on the second compression ramp. Although infrared thermography measurements returned a typical surface heat transfer distribution of boundary layer separation [77], no separation can be observed from the PIV
Figure 5.7: Average PIV (a), schlieren (b) and QIRT (c) results for the 15-30 wedge model

measurements. The same situation applies for the 15°-45° configuration discussed in the next section.
From the local value of the velocity, the Mach number can be computed under the assumption of adiabatic flow (constant total temperature), which is a justified approach for the present flow field if the boundary layer is excluded. The expression for the Mach number can be derived combining the definition for the Mach number $M$ and the energy conservation equation resulting in:
where $V$ is the velocity magnitude, $\gamma$ is the specific heat ratio, $R$ is the gas constant for air and $c_p$ is the specific heat at constant pressure. The subscript $\infty$ refers to the free-stream flow conditions. The flow in regions 1, 3 and 4 (see figure 5.6) is sufficiently uniform to allow comparison between the measured Mach number with inviscid compressible flow theory, which returns a good agreement (see Table 5.1).

### 5.3.1 Comparison with CFD computations

The measured flow field is compared to CFD computations, which were performed with LORE, a finite volume second order flow solver [90]. The turbulence model used in the computations is Menter’s shear stress transport (SST) model. Grid convergence has been established by performing computations on the baseline mesh consisting of 1 million cells and successively coarsening the grid by a factor of 2 and 4. Boundary layer transition was imposed at the first to second ramp junction, since for compression ramp models boundary layer transition is most likely to occur in the shear layer above the separated region or during the reattachment process [24]. Furthermore, the heat transfer measurements show that it is a transitional interaction (laminar at separation and turbulent at reattachment).

In figure 5.8 the synthetic and experimental schlieren images are given. They show good qualitative agreement with respect to shock wave position and shear layer separation location. However it is found that the separation shock has a larger angle in the computations with respect to the experiments resulting in a more upstream interaction with the leading edge shock. In the experiments it is found that the separation shock, reattachment shock and leading edge shock interact in more or less a single point. In case of the computational results the interaction of the reattachment shock with the leading edge shock occurs in approximately the same location.

A comparison between the heat transfer measurements and the CFD results is shown in figure 5.9 in terms of Stanton number (see section 4.4.5 for the denition of $c_h$). In the figure also the values computed from laminar boundary layer theory with the reference temperature correction [4] are plotted. It can be seen that on the front part of the model where the boundary layer is laminar, all three results are in good agreement. The separation point also agrees well, however in the separated region the measurements return slightly higher values that the CFD results. The heat transfer values on the second ramp are approximately 25% higher for the CFD results but the trend in the downstream development of the Stanton number is similar.
When comparing the computational results to the PIV data, it is also found that there is good qualitative agreement. However also in this case the separation region that resulted from the computations is larger where the separation region seems almost absent in the PIV measurements. This is due to a lack of measurement data near the model surface because of laser light reflections.
When the angle of the second ramp is increased from $30^\circ$ to $45^\circ$, the shock wave pattern over the second ramp changes significantly. Since the angle of the second ramp with respect to the oncoming flow exceeds the maximum value possible for an attached shock solution at Mach 7.5, a detached curved shock is formed instead of an oblique one. The shock structure is conjectured as shown in figure 5.12 in combination with the shock polar diagram. The schlieren image for the $15^\circ$-$45^\circ$ configuration (figure 5.13) shows a curved shock above the second compression ramp. The pattern is consistent with a variant of the Edney type V interaction.

Supported by the flow visualization, a theoretical description of the inviscid flow was made, where the separated region is modeled as an additional $4^\circ$ compression ramp. According to the corresponding shock polar diagram an oblique shock is formed.
Figure 5.13: Average PIV (a), schlieren (b) and QIRT (c) results for the 15-45 wedge model

at the intersection of the leading edge shock and curved shock, which is connected to the oblique reattachment shock by a Mach stem (strong shock solution). The theoretical (inviscid) pattern agrees with the experimental visualisation, albeit that the Mach stem is not very evident in the schlieren image. As a result of the complex interaction, at least two distinct regions are formed in the flow downstream: a region close to the wall where the flow has passed multiple oblique shocks (lower entropy loss), and a second region behind the curved shock. After subsequent expansion, the flow in the former region (region 4) develops into a wall-jet, with significantly higher momentum than the flow further away from the wall. This flow topology is found to be in agreement with Olejniczak et al. [61].
The heat transfer profile shows that the separation point has slightly moved upstream
with respect to the $15^\circ - 30^\circ$ model. At the second ramp the heat transfer strongly
increases exhibiting two local maxima of twice the intensity when compared to the
$15^\circ - 30^\circ$ case. The first maximum is associated to boundary layer reattachment,
while the second maximum is ascribed to the impinging shock (between regions 3
and 4) resulting from the interaction of the leading edge shock, curved shock and
reattachment shock. The peak values are followed by a sharp decrease due to the
expanding wall-jet flow. The increase towards the end of the model is ascribed to
flow compression resulting from the reflection of the shock causing the second heat
transfer peak.
Similar to the $15^\circ - 30^\circ$ case, the Mach number field was computed from the velocity
field. A straightforward comparison with theory is not possible since uniform flow
domains are difficult to discern due to the complex nature of the shock interaction.
Behind the leading edge shock the Mach number is measured to be $M = 4.6$ and this
is also found from theory. The low velocity region behind the curved shock complies
well with theory too (both measurement and theory give $M = 0.4$).

5.4.1 Comparison with CFD computations

Figure 5.14: Synthetic versus experimental schlieren image

The experimental results for the $15^\circ - 45^\circ$ ramp model are also compared to CFD
computations. Only for a fully turbulent flow field a fully converged computation was
obtained. The comparison between the experimental and synthetic schlieren image is given in figure 5.14. The overall shock locations (leading edge, separation and curved shock) are similar between computations and experiments. Although the separation point agrees well, the separation angle is much larger for the computations resulting in a larger separated region.

Also in this case the measured heat transfer was compared to theory (laminar boundary layer with reference temperature correction) and the agreement was similar to that found for the $15^\circ - 30^\circ$ model (figure 5.9) and therefore not plotted here. It was not possible to compare the measurements to the CFD computations, since in the measurements the boundary layer is clearly laminar while the computation was fully turbulent.

This is also apparent from the velocity fields (figures 5.15 and 5.16). Comparing the computed velocity field to the experiments also an overall qualitative agreement is found, apart from the separated region. The reason for the discrepancy between computations and experiments is believed to be due to three dimensional aspects of the flow field and the lack of proper turbulence modelling [84], [9]. Therefore a three dimensional flow survey is performed, the results of which are given in section 5.5.

### 5.4.2 Unsteady flow features and turbulence

The temporal fluctuations of the flow field over the double ramp were studied by means of high speed schlieren visualization. A high speed CMOS camera (LaVision HighSpeedStar V, 10 bit, 17 µ/pixel) was used at a frame-rate of 12 kHz having a
resolution of $512 \times 512$ pixels. The field of view was $16.7 \times 16.7 \text{ cm}^2$. The schlieren system had a continuous light source and the camera shutter time was set to $1/251000$ s. The schlieren images revealed an unsteady flow behaviour in both the separation and reattachment region. It is found that the leading edge shock remains stable while some movement of the separation shock and curved shock is detected. No correlation could be found between the movement of the separation shock and the curved shock. This is illustrated in figure 5.17 where the position of the separation shock changes while the curved shock remains in the same location. Additionally a small shock downstream of the separation shock is visualized (indicated by the arrow). In the upper picture where the separation shock occurs further downstream (separated region is smaller), this shock is stronger and further upstream than in the lower picture. Therefore, it is conjectured that the small shock can be associated with transition in the shear-layer, which decreases the size of the separated region and influences the position of the separation shock. Furthermore a movement of the curved shock was detected, however, compared to the movement of the separation shock it is small. This movement can be either a flow property due to the inherent instability of the interaction [12], [42] or it can be caused by the fact that the schlieren technique is an integrating line-of-sight technique.

Figure 5.17: Schlieren image showing the separation shock movement

The slip lines that emanate from the shock interaction cause large vorticity values which are shown in figure 5.18. When observing the corresponding PIV snapshots
downstream of the interaction on the second ramp, flow structures are visible as regions of decreased particle density (figure 5.19) at the location of high vorticity. The convection speed of these structures is evaluated by cross correlating the spatial distribution of the particle concentration in two PIV snapshots of an image pair. This results in a convective velocity of 590 m/s, which corresponds to the average velocity in the shear layer. Furthermore, the distance between the structures, as obtained from the PIV snapshot together with the measured convection velocity, enables to derive a characteristic frequency for the generation of these structures. From figure 5.19, the distance can be evaluated to be approximately 1.5 cm, which returns a characteristic frequency of 40 kHz.

Figure 5.18: In-plane vorticity as obtained from the PIV results

Figure 5.19: PIV snapshot of the flow over the second ramp

5.5 Three dimensional investigation of the flow over the 15-45 ramp model

Due to the presence of the curved shock over the second ramp and the occurrence of a large local subsonic region, three dimensional effects are expected to have a non-negligible effect on the nominally two dimensional flow configuration. For this reason an investigation of these three dimensional effects was performed by means of stereo PIV and QIRT, with oil flow as an additional surface flow visualization tool.

5.5.1 Experimental setup

An overview sketch of the setup is given in figure 5.20. The measurement planes are parallel to and located 20, 8, 4 and 0.5 mm above the surface of the first ramp. Also a plane parallel to the second ramp was investigated at a height of 1.5 mm. Figure 5.21 shows the measurement planes in relation to the relevant flow features. The two coordinate systems are aligned with the measurement planes, where the local y-axis is in spanwise direction and the z-axis is perpendicular to the measurement plane. The u, v and w velocity components are linked to the x, y and z direction respectively.
Two CCD frame straddling cameras are viewing the measurement plane from an angle in order to achieve a stereoscopic viewing effect. The cameras are located at an angle of 3° and 33° with respect to the measurement plane, this results in an angle of 30° between the two cameras. Usually an angle of approximately 40° is preferred [62] in view of the out-of-plane velocity component, however this was limited because of optical access issues.

![Figure 5.20: Setup of the stereo PIV system (top view)](image)

The effective field of view of the complete camera system is approximately 105 × 105 mm². The window size used for PIV processing was 32 × 32 pixels and an overlap factor of 75% was used. Effectively this results in a vector stepping of 0.5 mm. The laser sheet thickness in the experiments was approximately 1 mm. When referring to the position of the laser sheet, the distance with respect to the center of the sheet is meant. The measurement planes under investigation are indicated in figure 5.21.

![Figure 5.21: Location of the measurement planes and definition of the axis](image)

### 5.5.2 Outer flow region

First the outer flow regime is discussed, this is the part of the flow field outside of the region of separation. This will be done on the basis of the observations in the planes at $z_1 = [20, 8, 4]$ mm from the first ramp surface.

Figure 5.22 shows a particle image recording (a) in combination with the velocity field (b) for the plane at $z_1 = 20$ mm. In the free stream an out-of-plane velocity component is measured since the measurements plane is oriented at a 15° angle with respect to the free stream flow direction. Progressing downstream from the free stream the particle density increases due to the increase in thermodynamic density when crossing the strong curved shock. Moving further downstream, a region having a large spatial variation in particle density is recorded. This coincides with the location of the shear layer introducing large blob-like structures as is discussed in section 5.4.2. The location of the curved shock is also clearly visible from the velocity field (figure 5.22(b)) where an increase in the w-component is measured. When the measurement
plane intersects the shear layer the w-component increases again due to the higher momentum of the flow. The spanwise variation in the freestream flow field as seen from the streamlines is due to the divergence caused by the conical nozzle. Near the edges of the model the pressure difference over the side of the model causes a significant additional spanwise velocity component.

Progressing to the plane at \( z_1 = 8 \, \text{mm} \), the schlieren image in figure 5.21 shows that it intersects with the leading edge shock, separation shock and reattachment shock, however the leading edge shock lies outside the field of view of the PIV measurements. Behind the leading edge shock, the flow is parallel to the first ramp resulting in a region where there is no out-of-plane velocity component (figure 5.23(b)). Further downstream, in the raw PIV recording (figure 5.23(a)), the separation shock can be discerned by a slight increase of the particle density. Since the reattachment shock is much stronger, also a large increase of particle density is recorded. Although the separation shock is weak, the increase in the out-of-plane velocity component clearly indicates the position.

The measurement plane at \( z_1 = 4 \, \text{mm} \) from the surface cuts through the flow field where the boundary layer reattaches on the second ramp. From the PIV recording (figure 5.24(a)) the reattachment process is seen as a bright white region due to the local increase in particle density. Immediately downstream of this bright region a dark region is observed, this is due to PIV image preprocessing (minimum subtraction) to remove the laser light reflection. The position of the separation shock is indicated in figure 5.24(b) as observed from the increase in the out-of-plane velocity component. Since this is a weak shock, the \( u \) velocity component is only slightly affected. In the reattachment region the \( u \)-component decreases strongly to zero, see Figure 5.24(b). Overall may be concluded that the center part of the outer flow field is not influenced by edge effects, which are apparently limited to a distance of 20 \( \text{mm} \) from the sides of the model.
Figure 5.23: PIV results for the plane at 8 mm from the model surface

Figure 5.24: PIV results for the plane at 4 mm from the model surface
5.5.3 Inner flow region, first ramp

It appeared that the status of the leading edge geometry had a profound effect on the separated region. The wind tunnel model was initially manufactured having a sharp nose. Over time, due to its repeated use, small imperfections appeared on the leading edge. In order to investigate the effect of a leading edge that is less sensitive to imperfections, the nose was rounded by means of a strip of tape. The geometry of both leading edge configurations is shown in figure 5.25.

![Leading edge geometry: (left) with strip and (right) sharp nose](image)

Figure 5.25: Leading edge geometry: (left) with strip and (right) sharp nose

Because the largest three dimensional effects were expected to occur in the separated region, the flow was investigated in a plane at $z_1 = 0.5 \text{ mm}$ from the model surface. The result for the rounded leading edge is shown in figure 5.26. Close to the wall the separation shock is diffused into a compression region, which is visualized in figure 5.26(b) as the region where the velocity decreases to zero. In the raw PIV image (figure 5.26(a)) this is indicated by an increase in particle density. The streamlines in the separated region reveal a considerable spanwise velocity component, caused by the higher pressure in the separated region compared to the edge of the model. Although the separated shock is rather two dimensional, the flow is highly three dimensional and emptying the separated region thus effectively decreasing it.

The result for the sharp leading edge is given in figure 5.27(a). Comparing the interaction region in figures 5.26(b) and 5.27(b) it is found that the separation region is smaller for the sharp leading edge and that the separation shock is highly three dimensional. It was found that small imperfections in the leading edge caused longitudinal streaks to appear in the raw PIV recordings (figure 5.27(a)). The intensity of the streaks changes in spanwise direction and the presence has a profound effect on the extent of the separation. At $y_1 = 25 \text{mm}$ the separated region is reduced to zero and this is precisely the location where the streaks are observed in figure 5.27(a).

The effect of the leading edge imperfections was further investigated by means of infrared thermography. The heat transfer results that are reported in section 5.4 were obtained using the Agema 880 LWB system in line-scan mode (see section 4.3.1) on the center-line of the model. Using the CEDIP Titanium camera (see section 4.3.2) it was possible to obtain the heat flux distribution in both spatial directions. Figure 5.28 gives the heat flux distribution for both the rounded and sharp leading edge.
Figure 5.26: PIV results for the plane at 0.5 mm from the model surface with strip

Figure 5.27: PIV results for the plane at 0.5 mm from the model surface without strip
configurations. The position of the separation line is affected to a large extent by the leading edge geometry, as was also found from the PIV results. In case of the rounded leading edge the separation location is further upstream and more two-dimensional. The heat flux peak at the second ramp is present in both cases, for the sharp nose the maximum is higher and the location of the peak is shifted more upstream. The flow over the second ramp will be discussed more in detail in the next section.

In addition, it was found that when the Reynolds number is decreased for the sharp leading edge, the effect of the streaks on the separated region is decreased (figure 5.29). For the lowest Reynolds number the separation region is similar to that found for the rounded nose. This can be explained by the observation that at a lower Reynolds number, the relative strength of viscous forces increases causing a faster dissipation of the disturbances coming from the leading edge.

Figure 5.28: Overview of the QIRT results, $M_{\infty} = 7.5$ and $Re/m = 11 \times 10^6 [m^{-1}]$

Figure 5.29: Influence of the Reynolds number on the separation region for the sharp nose
In case of the rounded edge the effect of decreasing the Reynolds number (figure 5.30) was an upstream shift of the separation line and the reattachment line, effectively decreasing the separated region which is a behavior typical for transitional interaction where the boundary layer is laminar at separation and turbulent at reattachment [17], [37].

![Figure 5.30: Stanton number profiles, taken on the center-line of the model, for different Reynolds numbers (s is the distance along the model surface)](image)

5.5.4 Inner flow region, second ramp

It appeared that the leading edge geometry has limited effect on the heat transfer distribution on the second ramp. In case of the sharp leading edge the heat transfer peak on the second ramp is shifted upstream because the separated region is smaller. This is also apparent from oil flow visualizations, see figure 5.31, where the oil flow pattern on the second ramp is shown. The oil flow results show the upstream movement of the reattachment point when the sharp nose is compared to the rounded nose. In both cases the same streamwise streaks are observed as for the heat transfer results, which are thought to be the footprint of Görtler vortices that are formed due to the concave geometry of the separated region [69], [21]. A raw PIV recording in combination with the velocity field is given in figure 5.32. The PIV recording (figure 5.32(a)) reveals the Görtler vortices by the spatial variation in particle density. However, the effect on the velocity field is too small to be measured.

The velocity field shows the location of the reattachment region as the position where the \( u \)-component changes from negative to positive which coalesces with the first heat transfer peak; downstream of reattachment the flow accelerates. At the location of the second heat transfer peak the flow undergoes compression again. Finally, the flow expands over the remainder of the model which coincides with a streamwise decrease of the heat transfer.
Figure 5.31: Oil flow visualization

Figure 5.32: PIV results for the plane at 1.5 mm from the second ramp surface
Table 5.2: Mean and rms heat flux and Stanton number fluctuations at $x_2 = 15 \text{ mm}$, see figure 5.33

<table>
<thead>
<tr>
<th>$Re/m \times 10^{-6}$</th>
<th>$q \text{ [kW/m}^2\text{]}$</th>
<th>$c_h \times 10^3$</th>
<th>$\text{rms}(c_h) %$</th>
</tr>
</thead>
<tbody>
<tr>
<td>11.02</td>
<td>107.4</td>
<td>11.05</td>
<td>3.8</td>
</tr>
<tr>
<td>8.82</td>
<td>97.0</td>
<td>12.48</td>
<td>4.3</td>
</tr>
<tr>
<td>6.61</td>
<td>83.0</td>
<td>14.25</td>
<td>5.5</td>
</tr>
</tbody>
</table>

Figure 5.33: Profiles at $y = 15 \text{ mm}$

For the configuration with the rounded nose, a spanwise profile of the heat transfer was taken at the reattachment region for different Reynolds numbers at location $x_2 = 15 \text{ mm}$. The effect of the Görtler vortices on the heat transfer distribution is quite evident, see figure 5.33. In table 5.2 the mean heat flux and Stanton number are given at different Reynolds numbers. Rms fluctuations are found to be up to 5%. When the Reynolds number is increased the heat flux is found to increase while the Stanton number decreases, which is expected. The spatial fluctuations are found to be increased as the Reynolds number is decreased which is in agreement with previous investigations [20].

5.5.5 Flow field overview

A composition of several PIV measurement planes is given in Figure 5.34. It shows the apparent two dimensional flow structure in the center plane and the large separated region at the ramp junction, also the shear layer developing between the curved shock and the model surface is visualized. From this overview and the surface heat transfer maps in figure 5.28 a schematic of the flow field is established in figure 5.35.

A number of prominent flow features are represented as well as the streamlines at various locations. The separation line, reattachment line, shock impingement line and the curved shock are extended over the span of the model to show their variation in spanwise direction. At the centerline the overall shock structure is given including the leading edge shock and its interaction with the curved shock. The separation bubble is limited by the region between the separation and reattachment line. The flow inside
Figure 5.34: 3D impression of the flow over the double ramp

Figure 5.35: 3D schematic of the flow over the double compression ramp
the bubble is given at the centerline and near the edge of the model, where the finite span effect is apparent. Outside the separation bubble the flow is less influenced by the finite span and remains nearly uniform across the span. On the second ramp a striation pattern is indicated, which represents variations in the flow across the span caused by streamwise Görtler vortices in the flow. The overall flow features found in the investigation were remarkably similar to the result of CFD computations reported by Navarro-Martinez and Tutty [59].
Chapter 6

Roughness induced boundary layer transition

6.1 Introduction

Boundary layer transition is a crucial aspect in the design of the thermal protection system (TPS) of re-entry vehicles. The surface heat flux generated by a turbulent boundary layer can be up to three times higher compared to a laminar boundary layer under the same flow conditions.

The mechanisms through which boundary layer transition occurs are still not well understood, although progress has been made by using for example algebraic growth models [65]. Among the mechanisms promoting transition are Görtler instability, first and second mode instabilities [54] and crossflow instability. Transition may also be promoted through environmental disturbances by the receptivity mechanism. For example in ground based facilities the study of natural transition is often greatly influenced by noise being generated by the facility (noise from the wall turbulent boundary layer) [72]. Another important factor in promoting transition is surface roughness which is the specific aspect under attention here. In general the roughness type can be classified either two-dimensional or three-dimensional type and the roughness can be either isolated or distributed.

In the current study boundary layer transition due to the presence of an isolated roughness element is studied. The investigation was performed in the framework of the EXPERT project [55]. The front part of the Expert vehicle consists of two different types of material, the nose of the vehicle is produced out of C/SiC while the afterbody is constructed from metallic parts (PM1000 and Gamma-TiAl), see figure 6.1. Since both materials have different thermal expansion coefficients and will be subjected to different thermal loads, a step may occur in the vehicle geometry when the vehicle surface temperature increases during re-entry, see figure 6.2. In relation to this, one of the objectives of the investigation is to provide information on the maximum acceptable step dimensions so that transition is not promoted at Mach 9.

93
Several studies have been devoted in the past to formulate empirical correlations that describe boundary layer transition for hypersonic (re-entry) vehicles. In those studies, a transition parameter is correlated to a flow disturbance parameter. Transition is assumed to occur when, at a given value of the disturbance parameter, the transition parameter exceeds a critical value, see figure 6.3. For example, a typical transition parameter is the Reynolds number based on the boundary layer momentum thickness. Its critical value is that value above which vortices shed in the wake of the roughness element(s) that trigger the boundary layer instability [64].

In the present study three correlations are considered, the Shuttle correlation, the PANT correlation and the $Re_k$ correlation. In the next sections these correlations will be further explained.

### 6.1.1 Shuttle correlation

The shuttle correlation was developed to assess the location at which transition would occur on the windward side of the space shuttle orbiter [13] due to TPS tile misalignment, which means that it considers a three dimensional isolated roughness element. For the shuttle correlation the disturbance parameter was taken as the ratio between
the roughness element height $k$ and the local boundary layer thickness $\delta$. The transition parameter is given by the Reynolds number based on the momentum thickness divided by the Mach number at the local boundary layer edge. The correlation reads:

$$\frac{Re_{g, tr}}{M_e} = C \left( \frac{k}{\delta} \right)^{-1}. \quad (6.1)$$

The constant $C$ depends on the vehicle geometry under consideration and describes the location where effective transition occurs (for incipient transition the value of $C$ is lower). Since the shuttle correlation was developed for lifting-entry vehicles [64] it was also used in the development of the X-33. In case of the space shuttle orbiter geometry, the constant $C$ is equal to $C = 20$ for incipient transition and $C = 30$ for effective transition, while for the X-33 the constant for incipient transition is $C = 45$ and $C = 60$ for effective transition [89]. This difference seems to indicate that for a given state of the boundary layer ($Re_g$, $M_e$ and $\delta$) the boundary layer on the shuttle is easier disturbed by surface roughness with a given height than the boundary layer on the X-33.

### 6.1.2 PANT correlation

A correlation for hypersonic roughness induced transition on nose tips was developed in the Passive Nose-tip Technology program (PANT) [11] to study transition on ablative nose cones. Effectively it is a correlation which is suited for three-dimensional distributed roughness elements. The transition parameter is the Reynolds number based on the momentum thickness, while the disturbance parameter is the product of two ratios namely roughness height to momentum thickness and boundary layer edge temperature to wall temperature:

$$Re_{g, tr} = a \left( \frac{k}{\bar{T}_e} \right)^n. \quad (6.2)$$

The temperature ratio is included to take into account the (de)stabilising effect of temperature variations across the boundary layer. The coefficients $n$ and $a$ were obtained by applying curve fits to experimental data. Several choices have been used [11] notably $a = 500$ with $n = -1.5$ and $a = 215$ with $n = -0.7$ see also figure 6.10. In the transient growth analysis performed by Reshotko and Tumin [66] a similar correlation was found.

### 6.1.3 $Re_k$ correlation

As previously mentioned, roughness induced boundary layer transition is ascribed to the shedding of vortices from the top of the roughness elements triggering the transition [64]. This means that transition will occur at a certain value of the Reynolds number based on the roughness height ($Re_{kk} = \frac{U_k u_k k}{\mu_w}$, $\mu_w$ is used since the roughness elements are at wall temperature). Using this concept the location where transition occurs is that location where the local roughness Reynolds number exceeds the critical value. The critical roughness Reynolds number is determined a priori by combining
CFD and experiments. In this approach transition is detected experimentally and the corresponding \( Re_{kk} \) is determined by a CFD calculation to provide accurate values of \( \rho_k \) and \( u_k \). However, the calculation of \( Re_{kk} \) depends greatly on how accurate the boundary layer velocity profile is resolved, which makes it less convenient for design analysis purposes. For that reason the choice is made to take instead \( Re_\kappa = \frac{\rho_k u_k}{\mu_k} \) as the parameter to describe the sensitivity to transition, taking \( \rho \) and \( u \) at the boundary layer edge rather than at the roughness height.

Whereas both the PANT and shuttle correlation are conceived to be geometry independent, the \( Re_\kappa \) approach is only strictly valid for equivalent geometries. So for a given geometry, transition at a specific critical \( Re_\kappa \) is detected and this critical value can be used in extrapolation to real flight conditions.

### 6.2 Experimental setup and data reduction

The experiments have been performed in the Hypersonic Test Facility Delft (HTFD), see chapter 2. For this investigation, the tunnel was equipped with the conical M9 nozzle, producing a free stream Mach number \( M_\infty = 9.5 \). The unit Reynolds number was varied from \( 9.8 \times 10^6 \) to \( 19.7 \times 10^6 \) \( m^{-1} \) by changing the total pressure in the free stream between 44 and 88 bar with a total temperature of \( T_t = 585 \) K.

The wind tunnel model used in the experiments consists of a hemispherical nose (radius 5 cm) and a cylindrical after-body (length 10 cm), see figure 6.4 and in figure 6.5 a schlieren image of the flow around the model is shown. The model is manufactured of Makrolon, a polycarbonate material with favourable thermal properties for the application of QIRT which is discussed in section 2.3.2.

The roughness configurations studied in the experiments are a backward facing and forward facing step obtained with a relative shift of the nose with respect to the body axis, as well as separately attached elements like tripping wires (2D isolated roughness...

Figure 6.4: Model dimension in mm

Figure 6.5: Schlieren image of the flow around the model (\( h = -3.2 \ mm, \ M = 9.5 \) and \( Re/m = 19.7 \times 10^6 \ [m^{-1}] \))
element), sandpaper (3D distributed roughness) and patches of small spheres (grit, 3D isolated roughness). Step heights of $h = [-3.2, -1.0, 1.0, 3.2] \, mm$ were studied, where ‘-’ denotes a backward facing and ‘+’ denotes a forward facing step. The sandpaper grain size was varied between $k = 0.26 \, mm$ and $k = 0.98 \, mm$. To achieve a distributed roughness with a higher roughness height, spherical steel shot (grit) was used with a diameter of $k = 1.5 \, mm$ and $k = 2.0 \, mm$. The patches were applied directly downstream of the hemispherical nose and have a length of $20 \, mm$ (sandpaper) and $15 \, mm$ (grit) in flow direction. The tripping wires were glued to the model nose at different angles from the stagnation point $\theta = [30^\circ, 45^\circ, 60^\circ, 90^\circ]$, the wire diameter is $d = 1 \, mm$.

The IR system used to obtain the surface temperature variation consists of the Agema Thermovision 880 LWB infrared scanner with a BRUT data acquisition system, see chapter 4. The time evolution of the local surface temperature $T_s(s, t)$ in a one-dimensional semi-infinite solid is related to the local convective heat transfer $q_s(s)$ to the surface. Here, $s$ indicates the position on the model, measured as the distance along its surface. When $q_s(s)$ is assumed constant in time, the temperature evolution reads as (see section 4.4.2):

$$T_s(s, t) = T_i + 2q_s(s)\sqrt{\frac{t}{\pi \rho c k}},$$  \hspace{1cm} (6.3)

where $T_i$ is the initial (uniform) temperature of the model. From the above expression it follows that the ratio of the heat transfer relative to that at the stagnation point (where the maximum heat transfer is expected), can be obtained directly by normalizing the instantaneous temperature profiles:

$$\bar{T}(s, t) = \frac{T_s(s, t) - T_i}{T_{s,stag}(t) - T_i} = \frac{q_s(s)}{q_{s,stag}}.$$  \hspace{1cm} (6.4)

The validity of this approach was assessed by verifying that the temperature profiles obtained at different times were self-similar, which indicated that the normalized temperature function as defined above, is independent of time.

### 6.3 IR measurements

In figure 6.6 the heat transfer distribution is given for a smooth model and a model having either a backward or forward facing step at the nose-body junction. Here $s$ is the distance measured along the body, normalized by the total distance so that $s = 0$ at the stagnation point and $s = 1$ at the end of the model. At the stagnation point the normalized heat transfer has a value of one and decreases moving downstream due to boundary layer growth. In case of the forward facing step ($h = +3.2 \, mm$) a local decrease in heat transfer is observed upstream of the step caused by flow separation. The peak that follows is due to the exposure of the sharp step edge to the incoming flow. Further downstream the heat transfer approaches the heat transfer of the smooth model, see figure 6.6. For the backward facing step only a local decrease in heat transfer can be observed, which is ascribed to flow separation behind the step. From the re-attachment point onwards, the heat transfer profile follows the
profile obtained for the smooth model. From these observations it is concluded that even for this relatively large step height (3.2% of the model diameter), no significant heat transfer increase is induced and therefore it does not promote boundary layer transition at these conditions.

For the case of 3D distributed roughness (sandpaper, grit) the heat transfer measurements are depicted in figure 6.8 which shows only the region downstream of the element. The measurements show an increase in heat transfer for a roughness height larger than 0.5 \( mm \). When \( k < 0.5 \ mm \) no difference in heat transfer can be observed. If \( k = 0.74 \ mm \), the heat transfer is twice the laminar value, which is ascribed to turbulent mixing in the boundary layer. Since for the largest roughness heights the heat transfer still increases with roughness height it cannot be concluded that a completely turbulent boundary layer state is reached.

Applying steel shot (grit) on the model surface, an even higher roughness height
could be obtained and the heat transfer was found to increase further. At \( k = 2.0 \ mm \) the heat transfer has increased by a factor three with respect to the laminar case.

The heat transfer profiles for the 2D (wire) and 3D (grit-line) roughness lines are given in figure 6.9. Adding the wire to the model only causes a small increase in heat transfer with respect to the smooth model case. The increase is identical for all angular positions, except for the wire at \( 60^\circ \) (which seems to give little effect), the reason for this is unclear. When changing from a 2D (wire) to a 3D (grit-line) disturbance, the heat transfer increases significantly: compare the wire at \( 90^\circ \) with the grit line which is at the same position. Apparently, roughness with an 3D (azimuthal) varying roughness height is much more effective in perturbing the boundary layer.

### 6.4 Results and extrapolation to flight

The results from the current experiments are depicted in figure 6.10 in terms of the PANT correlation, figure 6.11 for the Shuttle correlation and figure 6.12 for the \( Re_k \) correlation. The values for the boundary layer parameters for the different roughness locations are obtained from a CFD calculation considering an isothermal wall \( (T_w = 290 \ K) \). The computation was performed with LORE, a finite volume second order Navier-Stokes solver [90]. The effective roughness location is taken as the most downstream position of the roughness element.

The experiments where a laminar boundary layer flow was observed (the heat transfer is approximately the same as that for a smooth model, \( q \approx q_{\text{smooth}} \)) are labelled with an 'L'. When a turbulent boundary layer was found, the label 'T' is used (the heat transfer is more than twice the value for a smooth model, \( q > 2q_{\text{smooth}} \)). In the intermediate case \( (q_{\text{smooth}} < q < 2q_{\text{smooth}}) \), the label 't' is used, suggesting transitional conditions.

#### 6.4.1 PANT and Shuttle correlation

From figures 6.10 and 6.11 it is clear that the 3D type of roughness introduces a behaviour in good agreement with the PANT and Shuttle correlations (labels relative to \( h = -1.0 \ mm, h = +1.0 \ mm \) and \( \phi = 90^\circ \) fall on top of each other, the same happens for \( h = -3.2 \ mm \) and \( h = +3.2 \ mm \)). The experiments performed below the critical disturbance parameter indeed returned laminar flow behaviour. Moreover, when the critical value (according to either the PANT or Shuttle correlation) is reached, the experiments returned transitional and turbulent conditions associated to increasing values of the heat transfer.

The situation for the 2D type of roughness is rather different. Basically no experiment returned evidence of turbulent flow conditions in the investigated range of the disturbance parameter, even although several of these cases exceed the transition prediction criteria significantly. It is therefore concluded that the 2D isolated roughness disturbs the incoming laminar boundary layer to a much lesser extent with respect to the equivalent (i.e. with the same roughness height) 3D type of distributed roughness. This is ascribed to the fact that the two-dimensional roughness elements introduce 2D disturbances, which should first develop into a 3D fluctuation before transition.
can be triggered. On the other hand, the 3D roughness elements directly produce 3D disturbances (stream-wise vortices), which affect flow mixing and trigger boundary layer transition more effectively.

Figure 6.10: Results compared with PANT correlation (PANT1: $n = -0.7$ & $a = 215$, PANT2: $n = -1.5$ & $a = 500$; sp = sandpaper, gr = grit and st = step)

Figure 6.11: Results compared with Shuttle correlation (sp = sandpaper, gr = grit and st = step)
6.4.2 \( Re_k \) correlation and extrapolation to flight

To determine the critical value of \( Re_k \) where transition occurs, \( Re_k \) is plotted versus the ratio of the heat transfer measured on a roughened model at a certain position and the heat transfer measured on a smooth model at the same position. The position where the heat transfer ratio is evaluated, is taken halfway the cylindrical after-body. The value of \( Re_k \) is calculated at the position of the roughness element. The results are plotted in figure 6.12.

![Re_k correlation diagram](image)

Figure 6.12: \( Re_k \) correlation diagram, shaded area depicts the critical \( Re_k \) for the Shuttle correlation using BL parameters obtained from a CFD calculation

Generally, starting from a smooth nose (laminar boundary layer) and increasing \( Re_k \) (increasing the roughness height), the heat transfer ratio will first remain approximately one and then start to increase with roughness height. This is caused by the fact that the boundary layer becomes increasingly more perturbed by the roughness element(s). Beyond a certain value of \( Re_k \) the heat transfer ratio is expected to remain approximately constant when \( Re_k \) further increases (turbulent plateau). When transposing the Shuttle correlation into the \( Re_k \) diagram using the boundary layer data obtained from the CFD calculation, transition for the current conditions is predicted at \( Re_k \approx 800 \) (see the shaded area in figure 6.12). So for \( Re_k < 800 \) a laminar boundary layer is expected, while for \( Re_k > 800 \) the boundary layer is expected to be turbulent. For the height of the plateau a heat transfer increase of a typical factor 3 is taken; this situation corresponds to a fully developed turbulent boundary layer (a turbulent flat plate boundary layer has approximately three times the heat transfer of the laminar boundary layer at given momentum thickness Reynolds number).

From figure 6.12 it may be observed that for 3D distributed roughness the heat transfer ratio increases from 1 to 2 across the critical \( Re_k \) value given by the Shuttle
correlation. This confirms again that the current experiments show good agreement with this empirical correlation. For the 3D isolated roughness elements this increase happens for a slightly $Re_k$. For the 2D isolated roughness the measured points are still in the laminar regime, indicating that the 2D elements do not induce boundary layer transition in the investigated range which extends to values of $Re_k$ 4 times the critical value for 3D. This clearly illustrates that 3D elements are much more effective in inducing boundary layer transition. In conclusion, the existing empirical correlations give reasonable predictions for transition by 3D roughness elements but not for 2D elements.

Finally, for equivalent geometries the critical roughness height Reynolds number concept may be used to evaluate the maximum roughness height allowed for ‘in-flight’ conditions. Assuming transition is described by applying the same critical $Re_k$ for both wind tunnel tests and real flight, the critical ‘in-flight’ roughness height can be calculated from the critical wind tunnel roughness height, correcting for the change in Reynolds number using the unit Reynolds number both in the wind tunnel and real flight. The critical ‘in-flight’ roughness height can be calculated as:

$$k_{flight} = k_{wind\ tunnel} \frac{Re/m_{wind\ tunnel}}{Re/m_{flight}}.$$  \hspace{1cm} (6.5)

In the current experiments the unit Reynolds number in the wind tunnel, $Re/m = 14 \times 10^6 \text{ m}^{-1}$, is twice the EXPERT ‘in-flight’ unit Reynolds number. As a consequence the critical roughness height must be multiplied by a factor two going from wind tunnel to flight conditions.
Conclusions

Experimental facility and measurement techniques
The hypersonic wind tunnel facility used in the present investigation is the HTFD, that operates according to the Ludwieg tube principle. The operation and flow conditions were carefully assessed using various measurement techniques. Pressure measurements were performed in the nozzle section in order to experimentally quantify the pressure losses in case a throttle nozzle has to be used. It appeared that the total pressure ratio in the throttle nozzle is 0.35 for the Mach 7 nozzle and this agrees well with the theoretical results. Furthermore, the boundary layer velocity profile was measured by means of PIV in the nozzle near the nozzle-test section junction. Using the Crocco-Busemann relation a displacement thickness of 11.4 mm was obtained, enabling to calculate the effective cross section to be used for Mach number determination. It was found that the Mach number at the nozzle exit was reduced from 7 to 6.7 after the boundary layer displacement correction. Due to the conical nozzle the flow keeps diverging downstream of the nozzle exit, resulting in a free stream Mach number of 7.5 in the test section. In order to confirm experimentally both the free stream Mach number and total pressure, static pressures were measured in the nozzle and pitot pressures were measured in the test section.
The free stream velocity field was provided by means of particle image velocimetry (PIV). For a storage tube temperature of 773 K a free stream velocity of 1033 m/s was measured with a uniformity of 0.2 % and a repeatability of 0.4 %. Since both the values of the Mach number and corresponding flow velocity are known, the static and total temperature could be deduced from the energy equation. In the current conditions a total temperature of 579 K and static temperature of 47 K resulted. These values agreed with Fay-Ridell stagnation point heat transfer measurements performed by means of quantitative infrared thermography (QIRT). The loss in total temperature from its storage tube level to that of the free stream is thought to occur in the fast acting valve section of the facility. Since the free stream stagnation temperatures are relatively low, the possibility of condensation was assessed. Up to a free stream Mach number of 7.5, the flow is above the equilibrium condensation limit. Up to free stream Mach numbers of 9.5 a supersaturated free stream may be expected, which
corresponds to the current operational experience of the HTFD.

PIV was implemented for use in the hypersonic facility and the seeding performance in high speed flow was assessed. For the solid TiO₂ seeding used in the experiments it was shown that ample light was available for illumination. Since particle slip is an important factor under these conditions in compressible flows its theoretical background is addressed. Furthermore, a method is outlined that enables to assess the particle performance by means of a shockwave test using an unbiased curve-fitting procedure. Solid titanium dioxide particles were tested having different crystal sizes of 30, 50, 170, 240 and 550 nm. As expected it was found that the slip length increases with particle crystal diameter. However, the slip length for the 30 nm particles was found to be larger than the 240 nm particles, which was ascribed to excessive particle agglomeration.

For steady compressible flows a procedure was provided that allows to compensate for the particle slip effects. The procedure was applied to the particle slip profiles that were measured to establish the relaxation length and it was shown that the region affected by particle slip could be reduced with 50%. Furthermore it was found that due to the procedure a shock shift of 0.3 nm appeared irrespective of the particle relaxation time. Finally the procedure was also applied to an experimental image, illustrating that the procedure only acts where large accelerations are measured and showing the large effect on the streamline pattern.

**Double compression ramp flows**

The flow over double compression ramps was studied by means of schlieren, QIRT and PIV. Two geometries were studied having 15°-30° and 15°-45° wedge angles. The flow field over the 15°-30° featured an Edney type VI interaction, the results from schlieren and PIV showed good agreement on both flow topology and shock angles. The Mach numbers determined from the PIV results, assuming adiabatic flow assumption (constant total temperature) also agreed well with theory. The surface heat transfer profile showed the classical features such as a gradual decrease due to laminar boundary layer development on the first ramp followed by a sharp decrease at the separation location. On the second ramp a peak in heat transfer was measured that was caused by flow reattachment. Finally, a comparison is made with CFD computations and a good agreement is found for a mixed laminar/turbulent boundary layer where transition is imposed at the ramp junction.

In case of the 15°-45° model an attached shock at the second ramp is not possible due to the large second ramp angle. Therefore, the flow features a curved shock which results in an Edney type V shock interaction. Again schlieren visualizations and the PIV results show good agreement as well as Mach number comparisons with theory. The surface heat transfer profile shows a double peak in the heat transfer distribution on the second ramp, the first peak is caused by boundary layer reattachment while the second peak is caused by the reflecting shock that emanates from the shock interaction. In the CFD computations good convergence was only obtained for fully turbulent flow and a comparison with PIV showed that the overall flow features were similar. However, the separated region was considerably smaller in case of the experiments. This may be caused by three-dimensional effects in the exper-
ments. Therefore, 3C Stereo-PIV was used to further investigate the flow field. It appeared that the overall flow structure was essentially two-dimensional, except near the edges. However inside the separated region a highly three-dimensional flow field was measured. It features large span-wise velocities that essentially empty the separation bubble, effectively shrinking it.

In due course of the experiments it was observed that the leading edge shape has a large effect on the flow in the separation region. Small imperfections in the leading edge delayed separation and at some locations the separation was even completely absent. Therefore a strip was added to the model nose in order to increase the leading edge radius and make the configuration more uniform. The effect was that separation was promoted and that the spanwise variations in the separation line were greatly reduced.

On the second ramp streamwise streaks were detected in the surface heat transfer distribution, oil flow pattern and raw PIV images which are associated with the presence of Görtler vortices. By increasing the Reynolds number, the overall heat transfer is found to increase and the Stanton number to decrease. Furthermore the spatial fluctuations in terms of relative rms Stanton number are found to be reduced with increasing Reynolds number.

**Roughness induced boundary layer transition**

The effect of roughness elements on the onset of boundary layer transition was investigated within the EXPERT framework. The state of the boundary layer is measured by means of infrared thermography. The wind tunnel model under consideration is a sphere-cylinder which is a simplification of the EXPERT geometry. The study focussed on the occurrence of boundary layer transition due to a backward or forward facing step. The surface step constitutes a two-dimensional isolated roughness element. Also other two-dimensional roughness elements in the form of wires that were positioned further upstream closer to the stagnation point were tested. Additionally three-dimensional isolated (grit-line) and distributed (sandpaper and grit-patch) roughness elements were applied and tested. The results were compared to the classical Shuttle and PANT engineering correlations for prediction of transition. Transition did not occur for the two-dimensional roughness elements and that the transition correlations could not be applied for these types of roughness. The three dimensional roughness elements were much more effective in triggering transition and a reasonable agreement was found with the shuttle and PANT correlations. Finally, an alternative simple transition correlation based on $Re_k$ was proposed, to allow for a straightforward extrapolation of transition prediction from wind tunnel to free flight flow conditions for a given configuration.
HTFD running time derivation

Figure A.1: $t, x$ diagram of the Ludwik tube operating principle

For the simple in figure A.1 wave we have [8]:

$$J^+ \text{is constant: } u + \frac{2a}{\gamma + 1} = \frac{2a_0}{\gamma + 1} \quad (A.1)$$

slope of $\Gamma^-$ in expansion fan: $u - a = \frac{x}{t} \quad (A.2)$

eliminating $a$ and $u$ respectively:

$$u = \frac{2}{\gamma + 1} \left( a_0 + \frac{x}{t} \right) \quad (A.3)$$

$$a = a_0 - \frac{\gamma - 1}{2} u \quad (A.4)$$

Along $\Gamma^+$ AS:

$$\frac{\text{d}x}{\text{d}t} = u + a = \frac{1}{\gamma + 1} \left( 4a_0 + (3 - \gamma) \frac{x}{t} \right) \quad (A.5)$$
with boundary conditions $x = -L$ and $t = \frac{L}{a_0}$. Solve differential equation of the type:

$$\frac{dx}{dt} = A + B \frac{x}{t}$$  \hspace{1cm} (A.6)

where in this case $A = \frac{4a_0}{\gamma+1}$ and $B = \frac{3-\gamma}{\gamma+1}$. The solution is of the type:

$$x = \alpha t + \beta t^k$$  \hspace{1cm} (A.7)

$$\frac{dx}{dt} = \alpha + \beta t^{k-1}$$  \hspace{1cm} (A.8)

$$= A + B (\alpha + \beta t^{k-1})$$  \hspace{1cm} (A.9)

Solving for $\alpha$:

$$\alpha = \frac{A}{1-B} = \frac{2a_0}{\gamma-1}$$  \hspace{1cm} (A.10)

and $k$

$$k = B = \frac{3-\gamma}{\gamma+1}$$  \hspace{1cm} (A.11)

So that:

$$x = \frac{2a_0}{\gamma-1} t + \beta t^{\frac{3-\gamma}{\gamma+1}}$$  \hspace{1cm} (A.12)

$\beta$ now follows from the boundary conditions $x_A = -L$, $t_A = \frac{L}{a_0}$:

$$\beta = -\frac{\gamma+1}{\gamma-1} L t_A^{\frac{3-\gamma}{\gamma+1}}$$  \hspace{1cm} (A.13)

Now the solution is complete:

$$\frac{x}{L} = \frac{2}{\gamma-1} \frac{t}{t_A} - \frac{\gamma+1}{\gamma-1} \left( \frac{t}{t_A} \right)^{\frac{3-\gamma}{\gamma+1}}$$  \hspace{1cm} (A.14)

Consider point $S$:

$$\frac{x_S}{t_S} = u_1 - a_1 = \frac{\gamma+1}{2} u_1 - a_0$$  \hspace{1cm} (A.15)

With $t_A = \frac{a_0}{L}$, this can be written as:

$$\frac{x_S}{L} = \frac{M_1 - 1}{1 + \frac{\gamma-1}{2} M_1 \frac{1}{t_A t_S}}$$  \hspace{1cm} (A.16)

From equation (A.14):

$$\frac{x_S}{L} = \frac{2}{\gamma-1} \frac{t_S}{t_A} - \frac{\gamma+1}{\gamma-1} \left( \frac{t_S}{t_A} \right)^{\frac{3-\gamma}{\gamma+1}}$$  \hspace{1cm} (A.17)

Combining equations (A.16) and (A.17) we can now eliminate $x_S$:

$$t_S = \frac{L}{a_0} \left( 1 + M_1 \right)^{\frac{\gamma+1}{2(\gamma+1)}}$$  \hspace{1cm} (A.18)
and
\[ x_S = L(M_1 - 1) \left(1 + \frac{\gamma - 1}{2} M_1\right)^{\frac{3 + \gamma}{3 \gamma - 1}} \] (A.19)

Now \( t_1 \) can be computed from:
\[ t_1 = t_S - \frac{dt}{dx} x_S = t_S - \frac{x_S}{u_1 + a_1} \] (A.20)

with
\[ u_1 + a_1 = \left(\frac{u_1}{a_0} + \frac{a_1}{a_0}\right) a_0 = \frac{M_1 + 1}{1 + \frac{\gamma - 1}{2} M_1} a_0 \] (A.21)

We finally get for \( t_1 \):
\[ t_1 = \frac{L}{a_0} \frac{2}{1 + M_1} \left(1 + \frac{\gamma - 1}{2} M_1\right)^{\frac{\gamma + 1}{2(\gamma - 1)}} \] (A.22)
Appendix B

Derivation of \( \frac{d_{\text{hot}}}{d_{\text{cold}}} = \left( \frac{T_{\text{cold}}}{T_{\text{hot}}} \right)^{\frac{1}{4}} \)

It is possible to eliminate the reflection of the expansion wave at the junction of the hot and cold tube by changing the cross section of both tubes. Consider the situation given in figure B.1. Here we have a cold tube (region 0) connected to a hot tube (region 1). Before the expansion wave passes both regions are at rest and the initial conditions are:

\[
p_1 = p_0 \\
T_1 < T_0
\]

When the expansion wave travels into the hot tube, the air in region 0 is accelerated to region 4. At a given moment, the expansion wave hits the temperature discontinuity and a reflected wave is formed as well as a contact discontinuity. Also the expansion wave travels further into the cold tube. Over the contact discontinuity the following must hold:

\[
p_{2'} = p_3 \\
u_{2'} = u_3
\]

In the case that we do not want any reflection then we need:

\[
p_3 = p_4 \\
u_3 = u_4
\]

The process when going from region 0 through 4 to 3 is described by the Poisson curve [8]:

\[
 u_{\text{post}} - u_{\text{pre}} = \frac{2}{\gamma - 1} \left\{ \left( \frac{p_{\text{post}}}{p_{\text{pre}}} \right)^{\frac{\gamma - 1}{2\gamma}} - 1 \right\} a_{\text{pre}} \tag{B.1}
\]

Expanding this relation for \( u_{\text{post}} \) results in:

\[
 p_{\text{post}} - p_{\text{pre}} = -p_{\text{pre}} a_{\text{pre}} u_{\text{post}} + k u_{\text{post}}^2 + O(u_{\text{post}}^3) \tag{B.2}
\]
where $k$ is a constant. Using the above equation in combination with the fact that $p_{2'} = p_3 = p_4$ and $u_{2'} = u_3 = u_4$ results in:

$$p_{2'} - p_0 = -\rho_0 a_0 u_{2'} + k_0 u_{2'}^2 + \mathcal{O}(u_{2'}^3) \quad (B.3)$$

The same can be done for the process between region 1 and region 2:

$$p_2 - p_1 = -\rho_1 a_1 u_2 + k_1 u_2^2 + \mathcal{O}(u_2^3) \quad (B.4)$$

Between region 2 and region 2' there is a steady 1D expansion. In that case the local Mach number (velocity) can be related to the pressure by:

$$\frac{p_{2'}}{p_2} = \left( \frac{1 + \frac{\gamma - 1}{2} M_2^2}{1 + \frac{\gamma - 1}{2} M_2'^2} \right)^{\frac{\gamma}{\gamma - 1}} \quad (B.5)$$

Also in this case an expansion can be made:

Steady quasi 1D \quad $2 \rightarrow 2'$ $p_{2'} - p_2 = -\rho_2 u_2 (u_{2'} - u_2) + \mathcal{O}(u_2^3) \quad (B.6)$

Isentropic flow between 1 and 2:

$$p_2 - p_1 = a_1^2 (\rho_2 - \rho_1) \quad (B.7)$$

Combining this with equation $(B.4)$:

$$\rho_2 = \rho_1 - \rho_1 \frac{u_2}{a_1} + \mathcal{O}(u_2^3) \quad (B.8)$$
Assume $u_2$ is a function of $u_2'$:

$$u_2 = c_1 u_2' + c_2 u_2'^2 + \mathcal{O}(u_2'^3)$$  \hspace{1cm} (B.9)

Now combine equations (B.3), (B.4) and (B.6):

$$p_0 - \rho_0 a_0 u_2' + k_0 u_2' - p_1 + \rho_1 a_1 u_2 - k_1 u_2^2 = -\rho_2 u_2 (u_2' - u_2) + \mathcal{O}(u_2'^3)$$  \hspace{1cm} (B.10)

Fill in equations (B.8) and (B.9) and the fact that $p_0 = p_1$:

$$-\rho_0 a_0 u_2' + k_0 u_2^2 + \rho_1 a_1 (c_1 u_2' + c_2 u_2'^2) = \ldots$$

$$-\rho_1 \left(1 - \frac{c_1 u_2'}{a_1} - \frac{c_2 u_2'^2}{a_1^2}\right) (c_1 u_2' + c_2 u_2'^2) (u_2' - c_1 u_2' - c_2 u_2'^2) + \mathcal{O}(u_2'^3)$$  \hspace{1cm} (B.11)

Combine all terms with $\mathcal{O}(u_2')$:

$$-\rho_0 a_0 + \rho_1 a_1 c_1 = 0$$

So now we get for $c_1$:

$$c_1 = \frac{\rho_0 a_0}{\rho_1 a_1} = \sqrt{\frac{T_1}{T_0}}$$  \hspace{1cm} (B.12)

Combine all terms with $\mathcal{O}(u_2'^2)$:

$$k_0 + \rho_1 a_1 c_2 - k_1 c_1^2 = -\rho_1 c_1 (1 - c_1)$$

And $c_2$ now becomes:

$$c_2 = \frac{-\rho_1 c_1 + \rho_1 c_1^2 + k_1 c_1^2 - k_0}{\rho_1 a_1}$$  \hspace{1cm} (B.13)

Fill in $c_1$ from equation (B.12):

$$c_2 = \frac{1}{a_0} \left( \sqrt{\frac{T_1}{T_0}} - 1 \right)$$  \hspace{1cm} (B.14)

So in the first order approximation:

$$\frac{u_2}{u_2'} = c_1 = \sqrt{\frac{T_1}{T_0}}$$  \hspace{1cm} (B.15)

Now consider the mass-flow between 2 and 2': $A_2 \rho_2 u_2 = A_{2'} \rho_{2'} u_{2'}$

$$\frac{A_{2'}}{A_2} = \frac{\rho_{2'} u_{2'}}{\rho_2 u_2}$$  \hspace{1cm} (B.16)

Isentropic process from 2 to 2':

$$p_{2'} - p_2 = a_2^2 (\rho_{2'} - \rho_2) = \gamma \rho_2 \left(\frac{\rho_{2'}}{\rho_2} - 1\right)$$  \hspace{1cm} (B.17)

with equation (B.6) this becomes:

$$-\rho_2 u_2 (u_2' - u_2) = \gamma \rho_2 \left(\frac{\rho_{2'}}{\rho_2} - 1\right)$$  \hspace{1cm} (B.18)
or
\[
\frac{\rho_{2'}}{\rho_2} = 1 - \frac{\rho_{2} u_2}{\gamma p_2} (u_{2'} - u_2) = 1 + \mathcal{O}(u_2^2)
\] (B.19)

The area ratio according to equation (B.16) now becomes:
\[
\frac{A_{2'}}{A_2} = (1 + \mathcal{O}(u_2^2)) \frac{u_2}{u_{2'}} = \frac{u_2}{u_{2'}} = \sqrt{\frac{T_1}{T_0}}
\] (B.20)

The area ratio can be expressed in terms of \(d_0\) and \(d_1\):
\[
\frac{A_{2'}}{A_2} = \frac{A_0}{A_1} = \frac{d_0^2}{d_1^2}
\] (B.21)

And now we can relate the cross section ratio to the temperature ratio in order to eliminate the wave reflection:
\[
\frac{d_0^2}{d_1^2} = \sqrt{\frac{T_1}{T_0}}
\] (B.22)
References


List of publications

Journal papers


Conference proceedings


13th Int. Space Planes and Hypersonic Systems and Technologies Conference. Capua, Italy: AIAA.


Acknowledgements

Als eerste wil ik graag mijn begeleiders, Fulvio en Bas, bedanken voor al hun advies en hulp tijdens mijn volwassen worden als onderzoeker. Fulvio, I would like to thank you for the trust that you have put in me during all the projects and conferences that we did together and I hope there will be many more in the future. Bas, dankjewel voor je zeer waardevolle bijdragen en al je hulp als ik weer eens vast zat. Willem, aan het begin van mijn promotie hadden we nog niet zo heel veel contact, maar dat is met de tijd wel gekomen. Ik weet niet zo goed hoe ik onze samenwerking moet omschrijven, mentor, vriend of collega, ik denk dat ze eigelijk allemaal wel kloppen. Ik ben blij dat ik met je mag samenwerken en hoop dit in de toekomst ook nog geruime tijd te gaan doen.

Daarnaast ben ik heel veel dank verschuldigd aan Eric, Frits, Peter en Nico. Zonder hen zouden er geen windtunnel modellen zijn gemaakt, geen experimenten zijn uitgevoerd en zou het aan de koffietafel een stuk minder gezellig zijn. Coltega phd’s bedankt voor de lunches met kroketten en soep, het was aangenaam om tussen de middag even met jullie uit het lab te ontsnappen en het zo nu en dan in de avond met een pintje voort te zetten.

My students, I would like to thank you all for your assistance, hard work and pleasant times in the lab. Ook wil ik graag mijn vrienden bedanken die me door mijn studie hebben geholpen: Tinus, Jenny, Johan, Kenneth, Michiel en Reinier. Ik ben blij dat ik jullie die ochtend op het eerstejaars weekend bij Delft CS ben tegengekomen.

Twee mensen die ik zeker niet mag vergeten zijn Gerrit en David. Dank jullie wel voor alle leuke (Agora) etentjes en pintojes waar we de (wetenschappelijke) wereld hebben verbeterd. David, elke keer als ik een krakende bank hoor moet ik weer glimlachen. Gerrit, bedankt voor: de koffie, aero-bbq’s, de seeding in de TST27, je experimentele expertise, de slechte muziek en het feit dat je mijn kamer-genoot was.

Ook mijn TUT vrienden mogen natuurlijk niet ontbreken. Jaap en Rosj, bedankt voor het in-, door-, bij- en na-pilzen in Nederland en de rest van Europa, het is toch altijd goed om bouwko’s als vrienden te hebben om over vliegtuigen te praten. Peter, jij ook bedankt voor de mooie tijd op de TU2, de Le-Mansjes, ring taxietjes, talloze pintojes (en katers) en alle andere dingen.

Mijn ouders mogen in dit rijtje natuurlijk niet ontbreken. Elize dankjewel voor het

This work was sponsored by the Dutch Technology Foundation STW under the VIDI - Innovation Impulse program, grant DLR.6198.

Ferry Schrijer
Delft, Januari 2010
Curriculum Vitae

Ferdinand (Ferry) Franciscus Johannus Schrijer was born on 14 July 1977 in Geleen, The Netherlands. He finished his secondary school (VWO) in 1996 after getting his HAVO diploma in 1994, both at Scholengemeenschap St Janscollege in Hoensbroek. In 1996 he started studying Aerospace Engineering at Delft University of Technology, from which he received his Master of Science degree in 2003. For his Masters thesis he worked at the Aerodynamics group on the application of infrared thermography to investigate a Mach 9 flow over a blunted cone-flare. During his thesis work he spent 3 months at the University of Naples to work on the application of infrared thermography.

In May 2003 he started his PhD under the supervision of Dr. F. Scarano and Dr. B.W. van Oudheusden. The research topic was the experimental investigation of flow phenomena pertinent to reentry aerodynamics such as shockwave boundary layer interaction and boundary layer transition. In his work he assessed the hypersonic wind tunnel used to perform the flow investigations and he applied particle image velocimetry in the hypersonic flow regime. The results of this work are presented in this thesis.

During his PhD work he was in the local organizing committee of two international workshops: "PIVNET II International Workshop on the application of PIV in Compressible Flows" in 2005 and the "EWA International workshop on Advanced Measurement Techniques in Aerodynamics" in 2008. Also he has been involved in two research projects: "ExStep Experimental investigation of surface-step induced boundary layer transition on the EXPERT vehicle" for ESA-ESTEC and the characterization of the flow field around transonic airfoils and their aerodynamic characteristics for the Northwestern Polytechnic University in China.

Furthermore he is a member of the CEAS Space Branch Aerothermodynamics Technical Committee

In 2007 he has been appointed as assistant professor at the Aerodynamics group of the Aerospace Engineering faculty at Delft University of Technology.