FLOW PROCESSES IN ROCKET ENGINE NOZZLES WITH FOCUS ON FLOW SEPARATION AND SIDE-LOADS

Jan Östlund

Licentiate Thesis
Stockholm, 2002

Royal Institute of Technology
Department of Mechanics
FLOW PROCESSES IN ROCKET ENGINE NOZZLES WITH FOCUS ON FLOW SEPARATION AND SIDE-LOADS

by

Jan Östlund

May 2002
Technical reports from
Royal Institute of Technology
Department of Mechanics
S-100 44 Stockholm, Sweden
ABSTRACT

The increasing demand for higher performance in rocket launchers promotes the development of nozzles with higher performance, which is basically achieved by increasing the expansion ratio. However, this may lead to flow separation and ensuing unstationary, asymmetric forces, so-called side-loads, which may present life-limiting constraints on both the nozzle itself and other engine components. Substantial gains can be made in the engine performance if this problem can be overcome, and hence different methods of separation control have been suggested, however none has so far been implemented in full scale, due to the uncertainties involved in modelling and predicting the flow phenomena involved.

The present thesis presents a comprehensive, up-to-date review of supersonic flow separation and side-loads in internal nozzle flows with ensuing side-loads. In addition to results available in the literature, it also contains previously unpublished material based on this author’s work, whose main contributions are:

(i) discovery the role of transition between different separation patterns for side-load generation,
(ii) experimental verification of side-loads due to aeroelastic effects and
(iii) contributions to the analysis and scaling of side-loads.

A physical description of turbulent shock wave boundary layer interactions is given, based on theoretical concepts, computational results and experimental observation. This is followed by an in-depth discussion of different approaches for predicting the phenomena. This includes methods for predicting shock-induced separation, models for predicting side-load levels and aeroelastic coupling effects. Examples are presented to illustrate the status of various methods, and their advantages and shortcomings are discussed. The third part of the thesis focuses on how to design sub-scale models that are able to capture the relevant physics of the full-scale rocket engine nozzle. Scaling laws like those presented in here are indispensable for extracting side-load correlations from sub-scale tests and applying them to full-scale nozzles.

The present work was performed at VAC’s Space Propulsion Division within the framework of European space cooperation.

Keywords: turbulent, boundary layer, shock wave, interaction, intermittent, overexpanded, rocket nozzle, flow separation, side-load, models, criteria, prediction, review.
# TABLE OF CONTENTS

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

2 NOZZLE FUNDAMENTALS ............................................................................................................................. 5

3 NOZZLE CONTOUR DESIGN AND FLOW FIELD .......................................................................................... 8
  3.1 INTRODUCTORY REMARKS ....................................................................................................................... 9
    3.1.1 Losses .................................................................................................................................................. 9
    3.1.2 Computational methods ...................................................................................................................... 9
    3.1.3 Initial expansion region ....................................................................................................................... 10
  3.2 CONICAL NOZZLES ..................................................................................................................................... 10
  3.3 IDEAL NOZZLE ........................................................................................................................................... 11
    3.3.1 Truncated Ideal Contoured nozzles (TIC) .......................................................................................... 13
    3.3.2 Compressed Truncated Ideal Contoured nozzles (CTIC) .................................................................. 14
  3.4 THRUST OPTIMISED CONTOURED NOZZLES (TOC) ............................................................................ 14
  3.5 PARABOLIC BELL NOZZLES (TOP) .......................................................................................................... 18
    3.5.1 Influence of skewed parabola design parameters on the flow field .................................................. 20
  3.6 DIRECTLY OPTIMISED NOZZLES ........................................................................................................... 22
  3.7 DESIGN CONSIDERATIONS OF CONVENTIONAL ROCKET NOZZLE ...................................................... 22

4 EXHAUST PLUME PATTERN ........................................................................................................................... 24

5 FUNDAMENTALS OF FLOW SEPARATION .................................................................................................... 27
  5.1 FLOW SEPARATION AS A BOUNDARY LAYER PHENOMENON ................................................................ 27
  5.2 SHOCK-WAVE BOUNDARY LAYER INTERACTIONS .................................................................................. 28
    5.2.1 The basic interactions ......................................................................................................................... 28
    5.2.2 The free interaction concept .............................................................................................................. 30
    1.1.3 The separation length ......................................................................................................................... 33
    1.1.4 Unsteadiness and 3-dimensional effects ............................................................................................. 34

6 FLOW SEPARATION IN ROCKET NOZZLES .................................................................................................... 40
  6.1 FREE SHOCK SEPARATION ....................................................................................................................... 40
  6.2 RESTRICTED SHOCK SEPARATION .......................................................................................................... 42
  6.3 CRITERA FOR FLOW SEPARATION PREDICTION IN ROCKET NOZZLES .................................................. 43
    6.3.1 Free shock separation criteria .......................................................................................................... 43
    1.1.2 Restricted shock separation criteria .................................................................................................... 51

7 MEASUREMENT OF FLOW SEPARATION AND SIDE LOADS ........................................................................ 53
  7.1 STATIC WALL PRESSURE MEASUREMENTS ............................................................................................... 53
  7.2 FLUCTUATING WALL PRESSURE MEASUREMENTS .................................................................................. 54
  7.3 SIDE LOAD MEASUREMENTS .................................................................................................................... 55
    7.3.1 Determination of the system frequency response function ............................................................... 58

8 SIDE-LOADS – PHYSICAL ORIGINS AND MODELS FOR PREDICTION ....................................................... 62
  8.1 SIDE-LOADS DUE TO TRANSITION IN SEPARATION PATTERN .................................................................. 62
    8.1.1 Origin of side load: observations of the VOLVO S1 nozzle flow ....................................................... 62
    8.1.2 Side-load model .................................................................................................................................. 66
  8.2 SIDE-LOADS DUE TO TILTED SEPARATION LINE .................................................................................... 68
  8.3 SIDE-LOADS DUE TO RANDOM PRESSURE PULSATION ......................................................................... 73
  8.4 SIDE-LOADS DUE TO AEROELASTIC COUPLING ..................................................................................... 79
    8.4.1 Aeroelastic analysis ............................................................................................................................. 79
    8.4.2 Experimental verification of the aeroelastic analysis ......................................................................... 85

9 FIELD MEASUREMENT TECHNIQUES IN OVEREXPANDED NOZZLES ..................................................... 89
  9.1 SHOCK VISUALISATION ............................................................................................................................ 89
  9.2 INFRARED CAMERA IMAGING .................................................................................................................. 91
  9.3 SEPARATION LINE VISUALISATION .......................................................................................................... 93
  9.4 FLOW VECTOR VISUALISATION OF EXHAUST PLUME FLOW ............................................................ 94
10 SCALING CONSIDERATIONS

10.1 AERODYNAMIC SCALING OF NOZZLE FLOWS WITH IDENTICAL GASES

10.2 AERODYNAMIC SCALING OF NOZZLE FLOWS WITH DIFFERENT GASES AND NOZZLE CONTOURS

10.3 S1 AND S3: TWO WAYS OF SCALING DOWN THE VULCAIN NOZZLE FLOW

11 REYNOLDS-AVERAGED NAVIER-STOKES CALCULATION METHODS

11.1 INTERACTIONS IN BASIC CONFIGURATIONS

11.2 REALIZABILITY CONSTRAINTS

11.3 OVEREXPANDED NOZZLE FLOW

11.3.1 Geometry and Mesh

11.3.2 Computational model

11.3.3 Comparison of computations with experiment

12 SUMMARY AND CONCLUSIONS

12.1 NOZZLE FLOWS WITH FREE SHOCK SEPARATION (FSS)

12.2 NOZZLES WITH TRANSITION OF SEPARATION PATTERN

12.3 AEROElastic EFFECTS

12.4 SCALING

13 REFERENCES

ACKNOWLEDGEMENTS
INTRODUCTION

The performance of rocket engines is highly dependent on the aerodynamic design of the expansion nozzle, the main design parameters being the contour shape and the area ratio. The optimal design of traditional bell-type nozzles for given operating conditions (i.e. chamber and ambient pressures) is already supported by accurate and validated tools. However, during operation at chamber pressures below design pressure, the flow will not be fully attached, but separated. The separation line will move towards the nozzle exit as the chamber pressure increases (during start-up) or when the ambient pressure decreases (during the vehicles ascent). Different kinds of dynamic loads occur in the nozzle when the flow is separated, the most well known of these being the so called side-load, that has attracted the attention of many researchers. This occurs during testing at sea level condition or during the first phase of the actual flight. The increasing demand for higher performance in rocket launchers promotes the development of nozzles with higher performance and hence larger area ratio, where the problem of flow separation and side-loads is present during a substantial part of the ascent.

One possible solution of the described problem is to adapt the nozzle contour during the flight to the changes of ambient and chamber pressure. Attempts in this direction, however, have not yet been successful due to weight and mechanical complexity of such adapting devices.

Another approach is to introduce so called Flow Separation Control Devices (FSCD), by which high area ratio nozzles can be operated at separated condition at high ambient sea level pressure without severe loads, thereby obtaining an improved overall performance. The feasibility of such devices is presently the objective of demonstration tests. The main reason why such devices do not yet exist in full scale is that several basic questions regarding the nature of the flow separation phenomena and corresponding side-loads remain to be answered, which means that basic research is needed.

Building of knowledge regarding flow separation and side-loads has been a continuous process at Volvo Aero Corporation (VAC) since 1993, when the Flow Separation Control working (FSC) group was formed with CNES, Snecma and Astrium.

VAC performed focused studies on the topic within the GSTP/FSC program, 1996-1999, under a contract with the European Space Agency (ESA) and the Swedish National Space Board (SNSB). This included sub-scale testing of rocket nozzles at the modified hypersonic wind tunnel HYP500 at the Aeronautical Research Institute of Sweden (FFA∗), in order to investigate the aerodynamic and aeroelastic behaviour of a parabolic contour with and without FSCD inserts.

In the subsequent FSCD-program since 1998, under contract with Swedish National Space Board (SNSB) and Centre National d’Études Spatiales (CNES), flow separation and side-loads have been studied analytically and experimentally in sub scale test campaigns. This work was performed in co-operation with FOI, CNES, Snecma, ONERA, LEA, DLR and Astrium. Within the frame of the FSCD-program, VAC performed new sub-scale nozzle tests at FFA’s test facility in Stockholm. In the FSCD program VAC has tested eight different nozzle concepts, which are listed in Table 1.

The present author has been actively involved in the VAC/FSCD activities since 1997, being in charge of the test design (including design of model contours), hardware set-up and instrumentation, as well as test logic and evaluation of test results. CFD-computations have been extensively used for designing the models. They are indispensable for a qualitative understanding of the physics and flow phenomena, and hence provide a necessary input for setting up model descriptions and making meaningful evaluations.

* is now a part of the Swedish Defence Research Agency (FOI)
This thesis presents a comprehensive, up-to-date review of turbulent shock wave boundary layer interactions in internal nozzle flows with ensuing side-loads, including results available in the literature as well as previously unpublished results. It gives a detailed physical description of the phenomena, based on theoretical concepts, computational results and experimental observation. This is followed by an in-depth review of different approaches for predicting the phenomena. This includes methods developed to predict shock-induced separation and models for prediction of side-load levels and aeroelastic coupling in rocket nozzles. Examples are presented to illustrate the status of various methods, and their advantages and shortcomings are discussed. The third part of the thesis focuses on the problems associated with designing sub-scale models that are able to capture the most relevant physics of the full-scale rocket nozzle. The presented scaling laws are indispensable for extracting side-load correlations from sub-scale tests and applying them to full-scale nozzles.

The main contributions of the author to the understanding and modelling of separation and side-loads concern
(i) discovering the role of transition between different separation patterns for side-load generation,
(ii) experimental verification of side-loads due to aeroelastic effects and
(iii) contributions to the analysis and scaling of side-loads.

It was observed already in the early 1970’s by Nave and Coffey that a transition in separation pattern from the free-shock separation (FSS) to the restricted shock separation (RSS) and vice-versa might occur. However, it was not understood that these transitions are the origin of two distinct side-load peaks, until Östlund et al presented the detailed analysis of the VOLVO S1 nozzle flow.

In highly aeroelastic cases a significant amplification of the side-loads can be obtained as the flow interacts with the mechanical structure. The study of aeroelastic effects in separated nozzle flows requires dynamic models of the mechanical nozzle-engine support system, the flow separation, as well as the coupling between these two. A simplified technique for handling these difficult coupling problems was proposed by Pekkari in the early 1990’s. Östlund made this model applicable by improving the aerodynamic modelling, which were subsequently verified in experiments.

In order to translate measured data into engineering correlations, it is necessary to relate model tests to the real rocket engine nozzles. Here, the main challenge is to reproduce the behaviour of the chemical reacting hot propellants using air with totally different gas properties. Some basic ideas on scaling of separation and side-loads are presented in this thesis, and their relevance to real rocket nozzles is discussed on the basis of two different sub-scale designs for Vulcain.
<table>
<thead>
<tr>
<th>Nozzle</th>
<th>Base Contour</th>
<th>e</th>
<th>Nozzle Picture</th>
<th>Description/Test objectives</th>
</tr>
</thead>
<tbody>
<tr>
<td>VolvoS1</td>
<td>Parabolic</td>
<td>20</td>
<td><img src="image1" alt="Image" /></td>
<td>This nozzle was designed with the geometrical definition of the Vulcan nozzle as a model. The primary objectives were to investigate the separation and side-load behaviour in a Vulcan like nozzle. More specific objectives were to study the influence of different structural response of the nozzle on the side load magnitude and investigate the degree of aerelastic coupling.</td>
</tr>
<tr>
<td>VolvoS2</td>
<td>Parabolic</td>
<td>20.8</td>
<td><img src="image2" alt="Image" /></td>
<td>S2 was dedicated to investigating the same h/w as S1 with an appended extension. The nozzle length was increased with approximately 25%. The extension was made in such a way that the pressure gradient was relatively high in the extension. The primary objective was to study the impact on the end-effect side load peak.</td>
</tr>
<tr>
<td>VolvoS3</td>
<td>Parabolic</td>
<td>18.2</td>
<td><img src="image3" alt="Image" /></td>
<td>This nozzle is a more refined scaling of the Vulcan nozzle compared with S1. The idea was here not only to duplicate the nozzle wall geometry, pressure and Mach number profile, but also to simulate the internal flow-field. As the chemistry is completely different between hydrogen / oxygen vs. air, it is impossible to get identical flow patterns. The contour was however made to have the same wall pressure profiles and Mach number distribution and the internal shock as close as possible to the Vulcan.</td>
</tr>
<tr>
<td>VolvoS4</td>
<td>Parabolic Polygon</td>
<td>18.2</td>
<td><img src="image4" alt="Image" /></td>
<td>The Polygon nozzle is a patented Volvo invention. The aim of the Polygon nozzle is to have a design with a side load reduction relative to a normal axisymmetric nozzle. The shape is three-dimensional, see the figure. This Polygon nozzle has an identical base-line contour as S3. The nozzle was made as an octagon with the polygonisation starting at the predicted position for the separation pattern transition. The objective was to evaluate the degree of side-load reduction with this concept.</td>
</tr>
<tr>
<td>VolvoS5</td>
<td>Parabolic contour (VolvoS5) + Positive pressure gradient on the second bell</td>
<td>18.2</td>
<td><img src="image5" alt="Image" /></td>
<td>S5 is a Dual-Bell nozzle, i.e., another FSG concept. This is a well-known nozzle type since several decades. Actual testing with separation has however been very limited and side load measurements were lacking when these tests were performed. The contour of S5 is equal to S3 in the first upstream section. This constitutes the first bell. The dual-bell contour used for this nozzle is then designed according to the principle of positive pressure gradient in the second bell. This means that the separation front in theory will travel directly from the start of the second bell out to the exit during the start transient.</td>
</tr>
<tr>
<td>VolvoS6</td>
<td>Truncated Ideal Contour (TIC)</td>
<td>20.7</td>
<td><img src="image6" alt="Image" /></td>
<td>S6 is a truncated ideal contoured nozzle, i.e., from a different family of contours compared with S1-89. This type of nozzle has no internal shock, why it only features free shock separation. The primary objectives were to investigate the separation and side load behaviour in this type of nozzle.</td>
</tr>
<tr>
<td>VolvoS6 short</td>
<td>Truncated Ideal Contour (TIC)</td>
<td>13.9</td>
<td><img src="image7" alt="Image" /></td>
<td>This is a shorter S6 nozzle. The objective was to investigate the influence of changes in the geometry downstream of the separation location and corresponding side-load.</td>
</tr>
<tr>
<td>VolvoS7</td>
<td>High Pressure Gradient (HPG)</td>
<td>24.6</td>
<td><img src="image8" alt="Image" /></td>
<td>In nozzles with an internal shock it exist a driving mechanism for transition between two different separation patterns. However, the contour can be designed such as this transition is suppressed. Hence, it will only be free shock separation in the nozzle. With the S7 nozzle this type of design was demonstrated.</td>
</tr>
<tr>
<td>VolvoS7 short</td>
<td>High Pressure Gradient (HPG)</td>
<td>20.3</td>
<td><img src="image9" alt="Image" /></td>
<td>This is a shorter version of the S7 nozzle. The objective was to investigate the influence of the downstream geometry on the separation and side-load.</td>
</tr>
<tr>
<td>VolvoS8</td>
<td>HPG with film injection</td>
<td>22.1</td>
<td><img src="image10" alt="Image" /></td>
<td>The S8 nozzle is a HPG contour with film injection. This nozzle was designed to have similar flow properties regarding mass flow rate and film injection pressure rate as the film cooled Vulcan 2 and Vulcan 2 nozzle. The objective was to study the impact of film injection on separation and side-load.</td>
</tr>
</tbody>
</table>

Table 1. Sub scale nozzles tested by VAC at FFA’s HYP500 facility.
The following papers are included in the Appendix:

**Paper 1**

**Paper 2**

**Paper 3**

**Paper 4**

The author has also contributed to the common FSCD group paper found in R 105, which have also been incorporated in parts in chapters 4 and 6.

In addition, the following work includes results reported by this author in numerous classified technical notes at VAC, ESA/ESTEC and CNES.

Results for which no references are given are previous unpublished results produced specifically for the purpose of the present report. They are mainly based on test results for the VOLVO S1, S3 and S6-S7 nozzles.
2 NOZZLE FUNDAMENTALS

The main system used for space propulsion is the rocket – a device that stores its own propellant mass and expels this mass at high velocity to provide force. This thrust is produced by the rocket engine, by accelerating the propellant mass particles to the desired velocity and direction, and the nozzle is that part of the rocket engine extending beyond the combustion chamber, see Figure 1. Typically, the combustion chamber is a constant diameter duct into which propellants are injected, mixed and burned. Its length is sufficient to allow complete combustion of the propellants before the nozzle accelerates the gas products. The nozzle is said to begin at the point where the chamber diameter begins to decrease. The flow area is first reduced giving a subsonic (Mach number < 1) acceleration of the gas. The area decreases until the minimum or throat area is reached. Here the gas velocity corresponds to a Mach number of one. Then the nozzle accelerates the flow supersonically (Mach number > 1) by providing a path of increasing flow area.

Simply stated, the nozzle uses the pressure generated in the combustion chamber, $p_c$, to increase thrust by accelerating the combustion gas to a high supersonic velocity. The nozzle exit velocity, $v_e$, that can be achieved is governed by the nozzle area ratio (i.e., the nozzle exit area, $A_e$, divided by the throat area, $A_t$) commonly called the expansion ratio, $\varepsilon$.

$$\varepsilon = \frac{A_e}{A_t} = \frac{1}{M_e} \left\{ \frac{2}{\gamma+1} \left(1 + \frac{\gamma-1}{2} M_e^2 \right)^{\frac{\gamma+1}{\gamma-1}} \right\}$$

Eq. 1
Where \( \gamma \) is the ratio of specific heat capacities.

The thrust, \( F \), produced by the nozzle can be expressed with some commonly used performance parameters in the propulsion community as:

\[
F = (\dot{m}v_e + p_e A_e) - p_s A_e = C_F p_e A_e = m I_{sp}
\]

Eq. 3

Where \( \dot{m} \) is the engine mass flow rate, \( C_F \) is the thrust coefficient (dimensionless) and \( I_{sp} \) the specific impulse \([m/s]\). \( v_e \) and \( p_e \) are average values of the velocity and pressure over the nozzle exit area.

\( C_F \) gives the amplification of the thrust due to the gas expansion in the rocket nozzle compared to the thrust that would have been obtained if the chamber pressure only acted over the throat area only.

\( I_{sp} \) is a measure of how efficiently a given flow rate of propellant is turned into thrust.

Using the isentropic relations the ideal specific impulse can be written as:

\[
I_{sp, ideal} = \frac{F}{\dot{m}} = v_e + A_e \frac{p_e - p_s}{m} = \frac{2\gamma RT_c}{\gamma - 1} \left[ 1 - \left( \frac{p_e}{p_c} \right)^{\frac{\gamma-1}{\gamma}} \right] + \frac{RT_c}{\gamma} \left[ \frac{2}{\gamma+1} \right] \cdot \varepsilon \frac{p_e - p_s}{p_e}
\]

Eq. 4

Here, \( T_c \) is the combustion chamber temperature and \( R \) is the gas constant.

Optimum performance is obtained with pressure matching (i.e. \( p_e = p_s \)) throughout the vehicle’s ascent. Inspection of equation 1, 2 and 4 indicates that this requires a variable \( \varepsilon \) i.e. an adaptable nozzle. However the required mechanism for such a nozzle is usually complex, heavy and difficult to cool and has therefore only been demonstrated in experimental rockets. Instead, a nozzle with a fixed expansion ratio is chosen as a compromise taking into consideration the performance requirement throughout the flight trajectory, see Figure 2.

Any off-design operation with either overexpanded or underexpanded exhaust flow induces performance losses. These inherent losses due to non-adapted flow condition for fixed geometry nozzles may rise up to 15%, compared to a continuously adapted exhaust flow.\(^9\) In principle, a first- or main stage rocket nozzle could be designed for a high area ratio in order to achieve high vacuum performance, but the flow would then separate inside the nozzle during low altitude operation, with an ensuing risk for side-load generation. The requirement of stable nozzle operation on ground together with high vacuum performance lead to the design of a nozzle that is highly overexpanded, but operationally full flowing at sea level condition, but significantly underexpanded at high altitude operation (\( p_e = 0 \)), where the main part of the flight trajectory takes place, and hence gives a low overall performance. This is illustrated in Figure 2, which compares the performance (specific impulse) during flight of an adapted ideal nozzle with an ideal and a real rocket nozzle (i.e. losses are included), both with fixed area ratio, \( \varepsilon = 45 \), as a function of altitude respectively. It can be seen that there is a great potential for performance increase if the negative effects associated with flow separation can be handled.
Figure 2. Performance and flow phenomena of a real nozzle versus altitude. (The model by Oates^10 has been used for modelling the decrease in atmospheric pressure with altitude.)
Different types of conventional convergent-divergent rocket nozzles exist, each producing their own specific internal flow field. Before analysing separation and side-loads in rocket nozzles, it is essential to understand the features of the different contour types, as the internal flow field determines the characteristics of the nozzle separation behaviour. Figure 3 shows examples of the Mach number distribution in some of the most common nozzle types. Methods to generate these nozzle contours will be discussed in the following.

Figure 3. Mach number distribution in a 15° conical ($\varepsilon=43.4$, $L=20.9$), TIC ($M_D=4.67$, $\varepsilon=43.4$, $L=17.7$), TOC ($\varepsilon=43.4$, $L=17.7$) and TOP ($\varepsilon=43.4$, $L=17.7$) nozzle (From top to bottom). The thick line indicates the approximate position of the internal shock.
3.1 INTRODUCTORY REMARKS

3.1.1 Losses

A real rocket nozzle is subject to different losses. The loss mechanisms fall into three categories: 1) geometric or divergence losses, 2) viscous drag losses and 3) chemical kinetic losses. Geometrical losses arise when a portion of the nozzle exit flow is directed away from the nozzle axis, resulting in a radial component of momentum. By calculating the momentum of the actual nozzle exit flow and comparing it to the ideal, parallel and uniform flow condition, the geometric efficiency, \( \eta_{geo} \), can be determined. This also includes profile losses due to a non-uniform velocity profile at the nozzle exit, e.g. caused by small recompression waves in the nozzle flow. By careful shaping of the nozzle wall contour, relatively high geometric efficiency can be obtained.

A drag force, produced by the viscous effects at the nozzle wall, acts opposite to the direction of the thrust, and therefore results in a decrease in nozzle efficiency. This viscous drag efficiency is defined as:

\[
\eta_{drag} = 1 - \frac{\Delta C_F_{drag}}{C_F_{ideal}} \tag{Eq. 5}
\]

where \( C_F \) is the thrust coefficient and \( \Delta C_F_{drag} \) is the difference in \( C_F \) due to viscosity. The drag force is obtained by calculation of the momentum deficit in the wall boundary layer. The third nozzle loss mechanism is due to finite-rate chemical kinetics. Ideally, the engine exhaust gas reaches chemical equilibrium at any point in the nozzle flow field, instantaneously adjusting to each new temperature and pressure condition. In real terms, however, the rapidly accelerating nozzle flow does not permit time for the gas to reach full chemical equilibrium. The chemical kinetics efficiency is calculated by comparing the one-dimensional kinetics (ODK) solution to the one-dimensional equilibrium (ODE) solution, or:

\[
\eta_{kin} = \frac{C_F_{ODK}}{C_F_{ODE}} \tag{Eq. 6}
\]

The combined effects of geometric loss, viscous drag and chemical kinetics then give the overall nozzle efficiency:

\[
\eta_{noz} = \eta_{geo} \eta_{kin} - (1 - \eta_{drag}) \tag{Eq. 7}
\]

Which typically varies between 0.90-0.98.

The optimum nozzle contour is a design compromise that results in a maximum overall nozzle efficiency. Experience tells the nozzle designer that a long nozzle is needed to maximise the geometric efficiency; but at the same time, nozzle drag and nozzle weight is reduced if the nozzle is shortened. If chemical kinetics is an issue, then acceleration of exhaust gases at the nozzle throat should be slowed down by increasing the radius of curvature of the throat region, at the cost of an increased nozzle length.

3.1.2 Computational methods

In supersonic flow the Euler equations are hyperbolic i.e. the flow is only determined by the upstream conditions. In this case the method of characteristics (MOC) can be used to calculate the nozzle flow field. This method is the most commonly used in the rocket nozzle society for generating nozzle contours and determining loads and performances. This method is described in any basic book about compressible flow see e.g. the classical book by Shapiro.\(^1\) This is e.g. the basic method used for wind tunnel design.

For the present work, Volvo in-house MOC codes for design of different types of nozzle contours is used.

For evaluation calculation, the Two-Dimensional Kinetics nozzle performance code (TDK) by Frey and Nickerson\(^2\) is the most commonly used and validated MOC program in the west. TDK can perform a
complete two-dimensional nozzle performance calculation including boundary layer and non-equilibrium chemical reactions. Thus, losses due to divergency, viscous drag and chemical kinetics effects are included.

### 3.1.3 Initial expansion region

In a nozzle the initial expansion occurs along contour TN, see Figure 4, and this determines the character of the downstream flow field. Choosing a corner expansion as the initial expansion TN yields a slightly shorter nozzle than the one obtained with a radius downstream of the throat for any given expansion ratio. However in rocket application a sharp corner downstream the throat are generally avoided due to chemical kinetics effects and a wall contour TN having a radius of curvature equal to 0.5 times the throat radius i.e. $r_{td} = 0.5r_t$, are widely used. Using a transonic-flow analysis, a constant Mach-number line TO can be defined at the throat. Given the flow condition along TO and the solid boundary TN, a kernel flow field TNKO can be generated with the method of characteristics. The flow in the kernel is entirely determined by the throat conditions and constitutes the expansion zone. This kernel is the basis in all MOC design methods.

![Figure 4. Initial expansion region, kernel.](image)

### 3.2 CONICAL NOZZLES

The conical nozzle, Figure 3 and Figure 5, is historically the most common contour for rocket engines since it is simple and usually easy to fabricate. There is a record of extensive nozzle research in the subject by the German scientists at Peenemünde. For the low area ratio nozzles considered for the V-2 rocket no significant advantage of using more complicated contours was found, probably due to the manufacturing reasons.

![Figure 5. Definition of conical nozzle.](image)

The exhaust velocity of a conical nozzle is essentially equal to the one-dimensional value corresponding to the expansion ratio, with the exception that the flow directions are not all axial. Hence, there is a performance loss due to the flow divergence. Assuming conical flow at the exit Malina[8][13] showed that the geometrical efficiency become:

$$\eta_{geo} = \frac{1 + \cos \alpha}{2}$$

Eq. 8

Where $\alpha$ denotes the nozzle cone half angle.
The length of the conical nozzle can be expressed as:

\[ L_{\text{cone}} = \frac{r_d (\sqrt{\varepsilon} - 1) + r_t (\sec \alpha - 1)}{\tan \alpha} \quad \text{Eq. 9} \]

Typically, cone half angles can range between 12° to 18°. A common compromise is a half angle of 15°. Due to its high divergence losses, the conical nozzle is nowadays mainly used for solid rocket boosters with small expansion ratios and small thrusters where simple fabrication methods are preferred. Nevertheless, a 15° conical nozzle is often used as a reference in comparing lengths and performance of other types of nozzles. A term used when designing bell nozzles is the “percent bell”. The phrase refers to the length of the nozzle compared to a 15° half-angle conical nozzle with the same \( \varepsilon \).

3.3 IDEAL NOZZLE

As mentioned above, the ideal nozzle is a nozzle that produces uniform exit flow conditions. The nozzle contour, which achieves this, can be designed with MOC. An outline of an ideal nozzle flow is shown in Figure 6.

Contour TNE is the diverging portion of the nozzle. After the initial expansion TN, the contour NE turns the flow over to axial direction. TN also defines the Mach number at K, which is equal to the design Mach number obtained at the exit. With the Mach line NK defined it is possible to construct the streamline between N and E with the use of MOC which patches the flow to become uniform and parallel at the exit and thus complete the nozzle design. In Figure 7 the left and right running characteristics are shown for an ideal nozzle. The design Mach number is \( M=4.6 \) and the gas properties are \( \gamma=1.2 \) with a molecular mass =13.63 g/mole. The TDK program was used to generate the starting line TO.

Figure 6. Basic flow structures in an ideal nozzle.

Figure 7. Left and right running characteristics for an ideal nozzle. \( M_{\text{Design}}=4.6, \gamma=1.2, L=50r_t \).
Figure 8. Ideal nozzle contours together with lines representing constant surface area, vacuum thrust coefficient and wall pressure respectively. (The 12 ideal contours have been generated with a Volvo in-house program and the performance have been calculated with TDK with boundary layer losses included)
3.3.1 Truncated Ideal Contoured nozzles (TIC)

The ideal nozzle is extremely long \((L=50r_t)\) in the specific case shown in Figure 7, and consequently is not practically feasible for rocket applications. The huge length is necessary to produce a one-dimensional exhaust profile. However, the thrust contribution of the last part of the contour is negligible due to the small wall slopes. A feasibly rocket nozzle can be obtained by truncating the contour, such contours are called truncated ideal contours (TIC). Ahlberg et al. proposed a graphical technique for selecting optimum nozzle contours from a family of TIC nozzles. The LR-115, Viking and the RD-0120 nozzle are examples of TIC nozzles. The method can be outlined as follows, a complete set of ideal nozzle contours is synthesised in a plot together with lines representing constant surface area, exit diameter, length and vacuum thrust coefficient respectively, see Figure 8. Within a given constraint such as expansion ratio (or exit diameter), surface area, or length an optimisation process can then be used to determine where to truncate the full nozzle contour to obtain maximum performance. The optimisation is best visualised by considering an enlarged section of the typical plot described above, see Figure 9. Point A in the figure, which is the point where the thrust coefficient line is tangent to a line of constant radius, \(r/r_t\), is the optima representing the exit point of a nozzle contour yielding maximum thrust for a given expansion ratio. Similarly, point B, the point which the thrust coefficient line is tangent to the constant surface area line, \(A_s/A_t\), representing the optima for a given surface area. Nozzle of maximum performance for a given length is represented by point C. Point D in the figure represents the most thrust obtainable from any given nozzle contour. When the contour extends beyond this point, wall friction becomes greater than the pressure forces giving a negative thrust contribution, which decreases the total performance. These sets of nozzles are not of practical interest, since the set of nozzles represented by point A has the same thrust but are smaller, i.e. shorter and smaller expansion ratio. As an example, the right and left running characteristics of a truncated ideal nozzle are shown in Figure 10 and corresponding Mach number distribution is shown in Figure 3. For this case the nozzle was optimised versus minimum surface area.

Nozzle contours for different design expansion ratios, \(\varepsilon_{D,i}\), or Mach No. \(M_{D,i}\).

Figure 9. Illustration of optimum nozzle for given constraints.
3.3.2 Compressed Truncated Ideal Contoured nozzles (CTIC)

In 1966 Gogish suggested a method to design extremely short nozzles. The basis of the method is to linearly compress a TIC nozzle. He suggested that such compressed truncated ideal contours (CTIC) or compressed truncated perfect contours (CTPC) as it is sometimes labelled, may have higher performance than a Rao nozzle (see next section) for the same envelope. A CTIC nozzle is obtained by linearly compressing of a TIC nozzle in the axial direction to obtain the desired nozzle length. A discontinuity in the nozzle slope produced in the above compression procedure is eliminated by a cubic equation which smoothly connects the linearly compressed curve with the initial circular curve. The above procedure yields a nozzle which has a more rapid initial expansion followed by a more severe turn back, as compared to the TIC nozzle. As a consequence, strong right-running compression waves will propagate from the compressed contour into the flow field. If the compression is strong enough, the characteristic lines will coalesce and form a right running oblique shock wave. The shock wave will increase the static pressure as the flow crosses the shock wave. If the shock wave lies near the nozzle wall, the pressure along the wall will be increased, thus increasing the nozzle thrust. This effect is the mechanism Gogish considered when he suggested that the compressed nozzle might yield higher performance than a Rao nozzle. However, as the study by Hoffman showed, this is not the case. Hoffman found that the Rao nozzle is superior to the CTIC nozzle. For some designs, however, the difference in performance was quite small indicating that an optimum CTIC nozzle is certainly a good propulsive nozzle. As an example the LE7A is probably a CTIC nozzle.

3.4 THRUST OPTIMISED CONTOURED NOZZLES (TOC)

A direct and elegant approach of designing nozzle contours is the method of using the calculus of variations. Guderley and Hantsch formulated the problem of finding the exit area and nozzle contour to produce the optimum thrust, for prescribed values of the nozzle length and the ambient pressure. However, the method was not widely adopted until the complicated solution method presented by Guderley and Hantsch was simplified significantly by Rao. Therefore the obtained nozzle contour is often labelled a Rao nozzle in the west. In Russia this nozzle type is better known as a Shmyglevsky nozzle since Shmyglevsky independently formulated the same method in Russia. The basic idea behind the method of generating a Rao-Shmyglevsky nozzle or the thrust optimised contour (TOC) as it sometimes is called are outlined in Figure 11. First, a kernel flow is generated with MOC, for a variety of $\theta_N$ and a given throat curvature $r_{td}$. For given design parameters (such as $\varepsilon$ and $M_E$ or $\varepsilon$ and $L$) the points P and N can now be found by satisfying the following conditions concurrently:

1. Mass flow across PE equals the mass flow across NP.
2. The resulting nozzle gives maximum thrust.
By using the calculus of variations, these conditions are formulated as specific relations that must be fulfilled along PE and NP see e.g. reference R 16.

Figure 11. Thrust optimised nozzle contour.

Once N and P are known, the kernel line TNKO is fixed, and the contour line NE is constructed in the following manner: By selecting points P', P'', etc. along line NK, a series of control surfaces P'E', P''E'', etc. can be generated to define E', E'', etc. along the contour NE.

In Figure 12 and Figure 13 the left and right running characteristics are shown for a Rao-Shmyglevsky nozzle. In Figure 3 the appertaining Mach number distribution can be seen. The design expansion ratio and length is $\varepsilon=43.4$ and $L=17.7r_i$ respectively and this nozzle has the same performance as the TIC nozzle presented in Figure 3 and Figure 10. The same initial conditions and gas properties as for the ideal nozzle design have been used.

It should be emphasised that the method produces a shock free flow in the region NPE governing the wall pressure. This is seen in Figure 13, where the characteristics do not cross each other. If point P is equal to point K, an ideal nozzle is produced by definitions. However, when $P \neq K$ a more drastic turning of the flow is obtained compared with an ideal nozzle, and compression waves formed in region NPE will coalesce into a right running shock downstream of the control surface PE, see Figure 3.

The thrust optimised contour has a significant increase in geometric efficiency compared with a 15° half-angle conical nozzle conical nozzle having the same expansion ratio see e.g. Huzel and Huang R 21. The corresponding length is in general between 80%-100% of the conical one. In Figure 14 a synthesis of the nozzle end points for 159 different Rao-Shmyglevsky contours with initial expansion angles between $\theta_N=25$-$34^\circ$ are shown. Superimposed on the plot of these end points are lines of constant surface area, constant vacuum thrust coefficient and exit wall pressure. The thrust coefficient includes both drag and geometrical losses and has been calculated using the program by Frey and Nickerson R 12. With the use of Figure 14, the designer get the first indication of the main dimensions of a TOC nozzle for a prescribed performance value, which can be used as a first approximation.
Figure 12. The left and right running characteristic lines in a Rao-Shmyglevsky (TOC) nozzle designed with a VOLVO in-house contouring program.

Figure 13. Close up of characteristic lines in the throat region in a Rao-Shmyglevsky nozzle.
Figure 14. Locus of optimum thrust nozzle exit points for various values of \( \theta_N \) together with lines of constant surface area, constant vacuum thrust coefficient and wall exit pressure.

As indicated in Figure 15, the shape of the TOC and the TIC nozzle are very similar. The main difference is that the TOC has a higher initial expansion followed by a more drastic turning of the flow compared with the TIC nozzle. This corresponds to a higher wall angle and Mach number downstream the throat and lower values of the angle and Mach number at the exit for the TOC compared with the TIC nozzle. This has no effect on the performance, however, the difference in flow structure will be seen to have large impact on the separation and side-load characteristics.

Figure 15. Comparison between an ideal truncated (TIC) nozzle and a Rao-Shmyglevsky (TOC) nozzle.
3.5 PARABOLIC BELL NOZZLES (TOP)

Since the computation leading to the Rao-Shmyglevsky nozzle is rather complicated and the resulting contour can only be described by a co-ordinate list, Rao proposed a skewed parabolic-geometry approximation to the Rao-Shmyglevsky nozzle contour from the inflection point to the nozzle exit:

\[
\left( \frac{r + b \cdot x}{r_i} \right)^2 + c \frac{x}{r_i} + d \frac{r}{r_i} + e = 0
\]

Eq. 10

These types of nozzles are often referred as Thrust Optimised Parabolic (TOP) nozzles. With a skewed parabola the nozzle contour is entirely defined by the five independent variables \(r_{td}, \theta_0, L, r_e, \text{ and } \theta_e\), see definitions in Figure 16a. With these independent variables a infinite number of contours can be generated. Selecting the proper inputs can approximate the Rao-Shmyglevsky (TOC) nozzle accurately without introducing any significant performance loss. Parabolic approximations to a number of Rao-Shmyglevsky contours are shown in Figure 16b-c.

These charts are used in the following manner:
1. Choose the length fraction \(L_f\) that gives the desired nozzle efficiency, Figure 16b
2. Knowing \(L_f\) and the desired expansion ratio the initial and final parabolic angles can be extracted from Figure 16c
3. Determine the nozzle configuration by using equation 10.

A common misunderstanding is that any parabolic bell nozzle of 80% length can always replace a 15° conical nozzle to yield increased performance. However, this is not a general truth. Rao examined nozzles with an expansion ratio of 100 and found that an arbitrarily chosen parabolic nozzle of 80% length only yielded 0.07% higher inviscid specific impulse than the conical one. He also showed that this parabolic contour could be replaced with a much smaller TOC nozzle, with the same length and performance but only 80% of the expansion ratio.

In Figure 3, Figure 17 and Figure 18 a parabolic-geometry approximation to the TOC nozzle in Figure 12 and Figure 13 is shown. The flow conditions along the wall are almost equal and, as expected, the performance is slightly less then the thrust optimised nozzle. There is however one main difference between the two nozzle flows. At the point N where the circular arc is continued with the parabolic curve there is a discontinuity in the contour curvature. This discontinuity generates compression waves that coalesce into an internal shock upstream the last left running characteristic line, i.e. the crossing of the right running characteristic lines in Figure 18. In a TOC nozzle this shock is formed downstream of the last left running characteristic line and hence has no influence of the wall pressure. In contrast in a TOP nozzle the internal shock appears upstream of this characteristic line, see the comparison between TOP and TOC nozzle in Figure 3, and hence affects the flow properties at the wall, given a slightly higher wall pressure at the nozzle exit. This feature of TOP nozzles has been proved to be useful for sea-level nozzles where a margin against flow separation is important. For this reason the Vulcain and SSME nozzles where designed with parabolic contour. Actually, the initial contour design of the SSME was a TOC. However, with this design the wall pressure at the exit would be about 31 % of the ambient pressure at sea level, i.e. in a range where past experience showed that nozzle flow separation is likely to occur. In order to avoid problems with flow separation, an additional margin in exit pressure was sought. This was done by performing a parametric study of different TOP contours, which resulted in a contour where the additional flow turning (and the accompanying internal shock) resulted in a pressure increase of 24% at the nozzle exit at a cost of only 0.1% in nozzle efficiency compared with the initial TOC design.
Figure 16. Performance and design data for a parabolic bell nozzle. a) the basic nozzle geometry; b) the geometric nozzle efficiency as function of percent length of an equivalent 15° conical nozzle; c) the initial and final parabolic angles versus desired nozzle expansion ratio (adapted from Rao et al.).
3.5.1 Influence of skewed parabola design parameters on the flow field

As mentioned before, a skewed parabolic nozzle contour is entirely defined by the five independent variables \( r_{td}, \theta_N, L, r_e, \) and \( \theta_E \), see definitions in Figure 16a. Each of this design parameter has its own influence on e.g. the Mach number distribution in the nozzle. This is illustrated by a series of calculation of TOP nozzle, see Figure 20.

The initial expansion angle \( \theta_N \) specifies the maximum Mach number that can be reached in the nozzle. Increasing this angle will increase the final kernel Mach number \( M_K \) and the wall Mach number \( M_N \) at the end of the expansion contour, see illustration in Figure 19 and Figure 20 c) and e). When increasing the value of the downstream wall throat radius of curvature, \( r_{td} \), the extension of the expansion contour and the length of the kernel will increase, Figure 20 c) and d) (The approximate location of the kernel is found by connecting the “knee”-points of the iso-M lines).

Figure 19. Illustration of kernel region OTNK. Flow inside the kernel is only determined of contour TN.
The nozzle length, $L$, gives the value of the Mach number on the centre line at the exit for given values of $R_{ad}$ and $\theta_c$. When the nozzle length corresponds to the length of the kernel the exit centre line Mach number will be identical to the final kernel Mach number $M_K$. With a shorter nozzle this value will be reduced. The nozzle length together with the exit radius, $r_e$, are the main parameters that define the average exit Mach number. The average Mach number and the wall Mach number at the exit will increase when increasing the length of the nozzle, see Figure 20 c) and f). Reducing the exit radius will of course reduce the average exit Mach number. However, no general truth can be said about the influence of the exit radius on the exit wall Mach number. For some cases a reduction of the exit radius will reduce the exit wall Mach number and in other cases increase it. The reason for this is that the contour is not only affected locally at the exit when changing the exit radius, but the entire contour will change. As can be seen in Figure 20 c) and g) a reduction of the exit radius for this special case will give a more severe compression close to the throat followed by a second expansion. This cause the nozzle C5, with a smaller area ratio compared to nozzle C1, to have a higher exit wall Mach number.
The wall pressure gradient and the wall Mach number distribution from the end of the expansion contour to the exit is governed by the nozzle length, exit radius and the nozzle exit lip angle, \( \theta_e \). Reducing the exit lip angle will increase the strength of the shock emanating from the inflection point at the end of the expansion contour. Increased shock strength will in turn result in an increase of the wall exit pressure and a reduction of the absolute value of the pressure gradient and the wall Mach number at the exit is achieved. The pressure gradient can even change sign if the exit lip angle is sufficiently reduced, as can be seen in the wall pressure profile for contour C6 in Figure 20 b).

3.6 DIRECTLY OPTIMISED NOZZLES

The classical design methods described above rely on an inviscid design. After an inviscid design has been completed, a boundary layer correction is added to compensate for the viscous effects. The main reason for calculating the inviscid and viscous flows separately was that the computational capability in the past was such that the Navier-Stokes (N-S) equations could not be used in the design of contours. Advances in the computational technology since the 1950's allow scientists nowadays to use N-S solvers in parallel with direct optimisation techniques in the design loop. A typical design or an optimisation may include the following steps:

1. The design requirements are specified.
2. An objective function is constructed. The minimum or maximum of which yields the design requirements, e.g. max. performance and min. nozzle weight etc.
3. The set of design parameters or variables is specified.
4. An initial value for each of the design parameters is estimated.
5. An initial solution is computed by using the estimated design parameters.
6. The objective function is computed from the difference between the design requirements and the computed solution.
7. The sensitivity of the objective function to the design parameters is calculated.
8. An optimisation problem is solved to generate a new set of design variables.
9. A new solution is computed and compared with the design requirements.
10. If the design requirements are met or a minimum or maximum is reached, then the procedure stops, otherwise the process is repeated from step 6 onward.

Since the resulting contour with this method deviates from an ideal contour, compression waves will be generated in the nozzle. These waves can in some cases converge and an internal shock is formed inside the nozzle in the same way as in parabolic or compressed truncated ideal nozzles.

Direct optimisation of nozzle contours takes into account the whole range of specific impulse losses unlike the other design methods described, and thus produces slightly better results. However, comparison has shown that the improvement in performance does not exceed 0.1% \( R^2_{25,R26} \). Hence, the choice of contouring method has thus little influence on the performance of conventional nozzles. This is however not the case for all rocket nozzles. For engines operating on metal-containing fuels (liquid or solid), high expansion ratio nozzles can at present only be contoured by direct optimisation methods, since the Rao-Smyglevsky or the Ahlberg method do not rule out the precipitation of metal oxide particles on nozzle walls, and the consequent loss of specific impulse, eroding and destroying the contour \( R^2_{27,R28} \). Another example where direct optimisation must be used is for low Reynolds number nozzles, since the classical approach with a boundary layer correction of an inviscid designed contour breaks down when the viscous effects are large, \( R^2_{29} \).

3.7 DESIGN CONSIDERATIONS OF CONVENTIONAL ROCKET NOZZLE

When designing a rocket nozzle the appropriate configuration is highly dependent on manufacturing methods, given limitations on the main dimensions, cooling requirements, the influence of the nozzle weight on overall rocket performance, etc. Detailed examination of all these aspects requires knowledge in several engineering field’s, not considered in this work. However, it should be pointed out that one of the most basic demands in the design loop of a real rocket nozzle is to keep the nozzle weight down. With increasing nozzle weight a number of problems arise. The nozzle will be more difficult to handle and fabricate. The loads and power required for gimbaling (vector control) and moving the engine increase, and thereby the weight and
complexity of the thrust vectoring system etc. As a result it seems reasonable to keep the nozzle length or surface area at a minimum. The main gas dynamic problem lies in optimally contouring the nozzles in order to minimise losses of efficiency and the main design methods have been outlined above. For the sake of simplicity the exhaust gases have here been assumed to expand adiabatically and behave like an ideal gas with a constant ratio of specific heats. Analysis of rocket nozzle flows in any real case should of course include radiative heat loss, chemical reactions due to incomplete combustion, and chemical properties of the exhaust gases, however these features do not alter the general methodology or results shown above.

It should also be mentioned that the choice of contour type is will depend upon the application, i.e. if the nozzle is to be used as an upper-stage, first-stage or booster nozzle etc. The TIC nozzle is the only rocket nozzle that produces a shock free flow, whereas in a conical, CTIC, TOC, TOP and the direct optimised nozzles an internal shock is formed inside the nozzle. In Figure 3 the difference of the internal flow field between the conical, TIC, TOC and TOP nozzle is illustrated. For first-stage nozzles, which operate from sea-level to high altitudes, this differences is essential since the internal shock has a strong influence on the global shock pattern of the exhaust plume and determine the flow separation shock pattern and the side load behaviour of the nozzle, see paragraph 6.2 and 8.1. If upper-stage engines are not used for stage separation there is no considerable flow separation at start up, hence the choice of contour has a much smaller importance.
Nozzles of high performance rocket engines in use for first- or main stage propulsion, e.g. the American SSME, the European Vulcain, or the Japanese LE-7, operate from sea-level with one bar ambient pressure up to near vacuum. At ground, these types of engines operate in an overexpanded flow condition with an ambient pressure higher than the nozzle exit pressure. As the ambient pressure decreases during ascent, the initially overexpanded exhaust flow, passes through a stage where it is adapted i.e. the ambient pressure is equal to the nozzle exit pressure, and then finally becomes underexpanded. Figure 21 and Figure 22 shows photographs of nozzle exhaust flows during these two types of off-design operation. At high altitudes, the underexpansion of the flow results in a further expansion of the exhaust gases behind the rocket as impressively illustrated in Figure 21 d), taken during a Saturn 1-B launch.

In the case of overexpanded flow, the exhaust flow adapts to the ambient through a system of oblique shocks and expansion waves, which leads to the characteristic barrel-like form of the exhaust plume. Different shock patterns in the plume of overexpanded rocket nozzles have been observed, the classical Mach disk, Figure 21 a), the cap-shock pattern, Figure 21 b) and the apparent regular shock reflection at the centreline, Figure 21 c). In ideal and TIC nozzles, a transition between Mach disc and the apparent regular shock reflection can be observed as the degree of overexpansion is decreased. This is because a nozzle flow with a small overexpansion can adapt to the ambient without forming a strong shock system, i.e. the Mach disc. In nozzles featuring an internal shock, e.g. TOC, TOP and CTIC nozzles, the cap-shock pattern can be observed. The difference between the Mach disc and cap-shock pattern is illustrated in Figure 22. Figure 21 b) proves the existence of the cap shock pattern in the exhaust plume of the Vulcain nozzle, which has a parabolic contour. This is the pattern first observed at the nozzle exit during start up. By increasing the combustion chamber pressure, the flow becomes less overexpanded. At some point the internal shock intersects the centreline and a transition to a Mach disc pattern takes place, see Figure 21 a) and Figure 23.

Recent sub-scale experiments performed within the European FSCD group also confirmed the stable existence of the cap shock pattern in the plume of parabolic sub-scale rocket nozzles.

---

* In case of axisymmetrical flow, a pure regular reflection at the centreline is not possible. Instead, a very small normal shock exists at the centreline.
Figure 22. Exhaust plume patterns for parabolic subscale nozzles, with cap-shock pattern, a) S1 VAC FFA, b) TOP ONERA, c) P6 TOP DLR, and d) for a truncated ideal nozzle, with Mach disk, P6 TIC DLR, also published in R 105.

Figure 22 a-c) show Schlieren images of the exhaust plume of parabolic sub-scale nozzles tested at DLR, ONERA, and FFA. For comparison, the exhaust plume of a truncated ideal nozzle is also shown where the classical Mach disk is clearly visible.\textsuperscript{R8}
Figure 23. Illustration of transition between cap shock and Mach disc pattern: The transition occurs when the normal shock hits the reflection point of the internal shock at the symmetry axis.

The above described shock patterns are not only an exhaust plume phenomenon. They also exist inside the nozzle at highly overexpanded flow conditions, when the jet is separated from the nozzle wall. As will be shown later in section 6 and 8, the different shock patterns determine the characteristics of the nozzle separation and side-loads.
5 FUNDAMENTALS OF FLOW SEPARATION

5.1 FLOW SEPARATION AS A BOUNDARY LAYER PHENOMENON

In 1904, Prandtl showed that flows with low friction in the vicinity of bodies can be subdivided into two regions: a thin layer close to the body, the so-called boundary layer (originally called friction layer due to the predominance of friction), and the remaining flow, the potential flow where friction effects can be neglected. In the boundary layer itself, the flow at the wall must follow a no-slip condition. Hence, the boundary layer is decelerated by the wall, but accelerated by the outer flow. The static pressure, constant across the boundary layer, is governed by the main flow.

In flows with favourable or zero wall pressure gradient, the boundary layer is attached to the wall. This can be different in the case of an adverse wall pressure gradient. If the wall pressure increases in the main flow direction, kinetic energy of the fluid particles is transformed into potential energy. However, fluid particles close to the wall only have a small kinetic energy because of their lower velocity. Therefore they are stopped by the pressure rise, and may be even forced to flow in the reverse direction. In this case the boundary layer is separated from the wall, and the recirculation region is developed in the vicinity of the wall.

Flow separation requires the existence of both friction and an adverse wall pressure gradient in a flow along a body. If one of these two conditions is suppressed, flow separation can be prevented. Prandtl proved this with different experiments, e. g. with a flow around rotating cylinders or with a diffuser with boundary layer suction. Also, flow separation might not occur if the adverse pressure gradient is weak. In this case, the normal exchange of momentum inside the boundary layer can be sufficient to transport momentum from the mean flow to the wall; consequently, the kinetic energy of the particles close to the wall can be high enough to withstand the pressure rise without separation. Turbulent boundary layers with their characteristic high lateral exchange of momentum therefore separate much later than laminar boundary layers, where the momentum transport only consists of molecular movements.

At the separation point of two-dimensional boundary layers, planar or axisymmetric, the wall shear stress becomes zero,

\[ \tau_w = \mu (\partial u / \partial y)_w = 0 \]  Eq. 11

From this equation, and the velocity profile, the behaviour of the derivatives of \( u \) in wall-normal direction can be estimated.

In order to get a closer understanding of the separation processes, the momentum equation in wall-parallel direction is considered. The chosen non-conservative formulation is valid for a Newtonian fluid in a Cartesian co-ordinate system, neglecting volumetric forces:

\[ \rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = \]  Eq. 12

\[ -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left[ 2\mu \left( \frac{\partial u}{\partial x} - \text{div} \ v \right) \right] + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right] + \frac{\partial}{\partial z} \left[ \mu \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) \right] \]

If an arbitrary point at the wall is considered, the non-slip condition yields \( u = v = w = 0 \) for all velocity components as well as for their derivatives with respect to time and to the wall-parallel directions \( x \) and \( z \). Substituting \( \text{div} \ v = \partial u / \partial x + \partial v / \partial y + \partial w / \partial z \), and by assuming a constant viscosity across the boundary layer, the expression is simplified as follows:

\[ 0 = -\frac{\partial p}{\partial x} - \frac{2}{3} \mu \cdot \frac{\partial}{\partial x} \left( \frac{\partial v}{\partial y} \right) + \mu \frac{\partial^2 u}{\partial y^2} \]  Eq. 13
Because of \( \partial \partial x (\partial v/\partial y) = \partial \partial y (\partial v/\partial x) \), the second term on the right side of Eq.13 becomes zero. Consequently, the following formula is valid for an arbitrary location at the wall:

\[
\mu \left( \frac{\partial^2 u}{\partial y^2} \right)_w = \frac{\partial p_w}{\partial x} \quad \text{or} \quad \frac{\partial \tau_w}{\partial y} = \frac{\partial p_w}{\partial x}
\]

Eq. 14

In this context, Eq. 14, was derived directly from the momentum equation, Eq. 12, only assuming a constant viscosity in the boundary layer, and is therefore valid for any point at the wall, including separation and recirculation zones. Schlichting \(^R\) \(^32\) derived Eq. 14 from the classic boundary layer equations, which only represent an approximation of the flow. The aforementioned derivation from the momentum equation shows that Eq. 14 is not only an approximation, but also an exact solution for the flow at the wall.

Eq. 11 can be used to show that \( (\partial^2 u/\partial y^2) > 0 \) at the separation point. Since the dynamic viscosity is always positive, Eq. 14 yields that in order to have separation the wall pressure gradient must be adverse:

\[
\left( \frac{\partial p_w}{\partial x} \right)_{s, \text{separation}} > 0
\]

Eq. 15

### 5.2 SHOCK-WAVE BOUNDARY LAYER INTERACTIONS

The above expressions are valid for subsonic as well supersonic flows. In the following we will however only discuss the case with turbulent supersonic flows, having an adverse pressure gradient of sufficient strength to cause the boundary layer to separate. When a supersonic flow is exposed to an adverse pressure gradient it adapts to the higher-pressure level by means of a shock wave system. Basically, separation occurs when the turbulent boundary layer cannot negotiate the adverse gradient imposed upon it by the inviscid outer flow. Thus, flow separation in any supersonic flow is a process involving complex shock wave boundary layer interactions.

#### 5.2.1 The basic interactions

The shock wave boundary layer interaction has been extensively studied in the last fifty years with the help of basic experiments, see e.g. references \(^R\) \(^33\)-\(^R\) \(^73\). The three basic configurations involving interaction between a shock wave and a boundary layer in supersonic flows are schematically represented in Figure 24. In all of these cases, the incoming outer flow is uniform flow streaming along a flat plate.

The first and conceptually most simple configuration is the wedge (or ramp) flow. Here, a discontinuity in the wall direction is the origin of a shock wave through which the supersonic flow undergoes a deflection equal to the ramp angle \( \alpha \), Figure 24 a).

The second type of flow is associated with the impingement on the wall of an incident oblique shock, which cause a deflection of the incoming flow, see Figure 24 b). The necessity for the downstream flow to be parallel again to the wall causes the formation of a reflecting shock issuing from the impingement point.

The third flow is induced by a step of height \( h \) facing the incoming flow, see Figure 24 c). Such an obstacle provokes separation of the flow at point \( S \). The rapid pressure rise accompanying separation gives rise to a shock wave emanating from a place very close to the separation point \( S \), and a separated zone develops between the separation point \( S \) and the step.
Figure 24. Basic shock/boundary layer interactions in supersonic flow. a) Ramp flow. b) Shock reflection. c) Step induced separation, adopted from R 77.

It has been shown in many experiments, that the major part of the shock / boundary layer interaction properties are nearly independent of the cause having induced the separation, whether being either a solid obstacle or an incident shock wave R 7, R 35, R 77. In fact the features of the static wall pressure for the above different experimental configurations are the same, see Figure 25. The wall pressure has a steep rise shortly after the beginning of the interaction at $I$. The flow separates from the wall at $S$, located a distance $L_s$ from $I$. The wall pressure then gradually approaches a plateau with almost constant pressure, labelled the plateau pressure $p_p$. The extent of this plateau reflects the size of the closed recirculation bubble and $p_p$ thus corresponds to the wall pressure in the bubble. A second pressure rise can be observed as the reattachment point at $R$ is approached. These characteristics are independent on the downstream geometry, as already mentioned, everything happens as if the flow were entirely determined by its properties at the onset of the interaction. This observation led Chapman et al. R 35 to formulate the free interaction concept.

Figure 25. Typical static wall pressure distribution observed in ramp, shock reflection and step flow; adopted from R 32 and R 48.
5.2.2 The free interaction concept

Chapman considered flow separation caused by the interaction between the boundary layer formed in a plane, adiabatic, supersonic uniform flow and a shock wave. The Mach number $M_i$ and the pressure $p_i$ define the inviscid uniform flow. The skin friction coefficient ($C_f$), the displacement thickness ($\delta^*$) etc. define the local characteristics of the boundary layer. The deflection angle of the mean flow in the streamwise direction is given by $\theta$, see Figure 26.

![Figure 26. Flow separation in uniform flow, notations.](image)

Chapman then made two assumptions about the flow in the interaction domain:

1. The flow structure follows a law of similarity
2. The deviation of the external non-viscid flow correspond precisely to the displacement effect of the boundary layer, i.e.,

$$\frac{d\delta^*}{dx} = \theta - \theta_i$$  \hspace{1cm} \text{Eq. 16}

By normalising the abscissa with an appropriate length scale $l$ characterising the extent of the domain, and the displacement thickness $\delta^*$ with the value at the origin of the interaction, $\delta^*_i$, one obtains:

$$\theta - \theta_i = \frac{\delta^*_i}{l} \int f_i(s) ds$$  \hspace{1cm} \text{Eq. 17}

Where $s = \frac{x-x_i}{l}$ and $f_i(s)$ is a non-dimensional function characterising the outer streamline deflection.

Integrating the simplified boundary layer momentum equation at the wall, Eq. 14, from $x=x_i$, after making it non-dimensional by introducing the wall friction $\tau_w = \frac{1}{2} \rho_i u_i^2 C_{f\beta}$ at $x=x_i$, results in:

$$\frac{p-p_i}{q_i} = \frac{l}{\delta^*_i} C_{f\beta} \int_0^x \frac{\partial (\tau_w/\tau_w)}{\partial (y/\delta^*)} ds = \frac{l}{\delta^*_i} C_{f\beta} f_2(s),$$  \hspace{1cm} \text{Eq. 18}

where $q_i = \frac{1}{2} \rho_i u_i^2 = \frac{1}{2} p_i \gamma M_i^2$

$f_2(s)$ is a new dimensionless function characterising the pressure rise.

By multiplying Eq. 17 by Eq. 18, $\frac{l}{\delta^*_i}$, is eliminated and one obtains:

$$F(s) = \sqrt{f_1 \cdot f_2} = \sqrt{\frac{p-p_i v(M_i)-v(M)}{q_i C_{f\beta}}}$$  \hspace{1cm} \text{Eq. 19}
Where \( \theta - \theta_i = v(M_i) - v(M) \) according to the Prandtl-Mayer law. Chapman then expressed the variation of \( v(M_i) - v(M) \) as function of \( \frac{p-p_i}{q_i} \), linearised for small pressure changes \( p-p_i \) (see e.g. Shapiro \(^{11} \) p. 436) and finally obtained:

\[
F(s) = \frac{p-p_i}{q_i} \sqrt{\frac{M_i^2 - 1}{2C_{f_i}}} 
\]

Eq. 20

The function \( F(s) \) is assumed to be a universal function, independent of Mach number and Reynolds numbers, to be determined from experiments. Figure 27 shows the generalised wall pressure correlation function \( F(s) \) obtained by Erdos and Pallone\(^{R7,8} \). The axial distance from the onset of the interaction has been normalised with the separation length i.e., \( l=L_s=x_s-x_i \). In the original work by Erdos and Pallone the distance to the pressure plateau of the extended separated flow was used as the characteristic length scale i.e., \( l=L_p=x_p-x_i \). From the figure following particular values of \( F \) can be found, \( F_s=F(1)=4.22 \) at the separation point and \( F_p=F(4)=6.00 \) at the plateau point.

The characteristic length \( l \) may be obtained by dividing Eq. 18 by Eq. 17, which gives:

\[
\frac{l}{\delta^*_i} = \frac{F(s)}{v(s=0) - v(s)} \sqrt{\frac{f_i(s)}{f_j(s)}}. \]

At the separation point (\( s=1 \)), this relation can be evaluated as:

\[
\frac{L_s}{\delta^*_i} = k \frac{F_s}{v_i - v_s} = \{\text{linearised}\} = k \sqrt{\frac{2}{C_{f_i} \sqrt{M_i^2 - 1}}} \]

Eq. 21

From different experiments an average value of \( k=0.37 \) has been obtained\(^{R7,7} \). However, the experimental data have a significant scatter around this value, \( k=0.37 \pm 0.2 \), presumably due to the difficulty of accurately determining the separation length, which in turbulent flows is very short.

The free interaction theory can be used to establish separation criteria for supersonic flow. The best known is the type of criteria first proposed by Erdos and Pallone\(^{R7,8} \) 1962. They determined the critical pressure rise.
between the pressure $p_r$ at location $s=r$ and $p_i$ ($s=0$) by assuming that the separation occurs when the pressure jump $p_r/p_i$ is:

$$\frac{p_r}{p_i} = 1 + F_r \gamma M_i^2 \left( \frac{C_0}{2 \sqrt{M_i^2 - 1}} \right)$$  \hspace{1cm} \text{Eq. 22}

This equation is obtained by rewriting equation 20 and using the expression for the dynamic pressure given in equation 18.

The pressure rise, corresponding to a “true” incipient separation case is obtained with $F_r=F_i=4.22$ and with $F_i=F_r=6.0$ (the value used by Erdos and Pallone) the “effective” incipient separation condition is obtained. The latter is observed when the separation bubble has reached a size large enough to produce a significant change in the flow field, while the former corresponds to the first appearance of a tiny separation bubble. The “effective” incipient separation condition, i.e. $F_r=6.0$, is the case, which is the most important for practical applications. Figure 28 shows the separation pressure rise for these two cases.

![Figure 28. Separation criterion deduced from the free interaction theory for uniform flow.](image)

From Figure 28, we can see some typical results of the pressure rise at the separation ($p_r/p_i$ or $p_r/p_i$) obtained with the free interaction theory:

- The pressure rise increases when the Mach number is increased.
- The pressure rise decreases when the skin friction coefficient decreases (corresponding to an increase of the Reynolds number).

Both of these tendencies have been confirmed by experiments performed at low to moderate Reynolds numbers and the criterion in equation 22 correlate experimental data well. However, in several experiments performed at high Reynolds numbers ($Re_{\delta_i}\times 10^5$) it has been observed that the pressure rise ($p_r/p_i$ or $p_r/p_i$) tends to become independent of the Reynolds numbers and even to slightly increase with it. As an example, Zukoski\textsuperscript{45} made a series of experiments on step flows at $Re_{\delta_i}\times 10^5$ with $M_i$ varying between 1.4-6.0, and found that the pressure rise at high Reynolds numbers depended only of the upstream Mach number $M_i$ as:

$$\frac{p_r}{p_i} = 1 + 0.73 \frac{M_i}{2}$$  \hspace{1cm} \text{Eq. 23}
This change of the influence of the Reynolds number as it becomes sufficiently high restricts criteria based on the free interaction theory to the range $Re_{\delta} < 10^5$ and Mach numbers $M_i < 5$

5.2.3 The separation length

The Reynolds number ($Re_{\delta}$), the displacement thickness ($\delta^*$) or the boundary layer thickness ($\delta$) are important viscous parameters that define the separation length $L_s$, i.e. the distance between the point where the wall pressure starts to rise to the point where the flow actually separates. Experiments on ramp flows have shown that in turbulent flow the separation length is very short, $L_s/\delta$ is of order 1, compared to the laminar case where the separation length is far larger than the incoming boundary layer thickness. For turbulent flow the influence of the Reynolds number on the separation length can be divided in two regions. For low or moderate Reynolds number ($Re_{\delta} < 10^5$) $L_s$ increases with increasing Reynolds number (see Figure 29a), in agreement with the free interaction theory. Whereas at high Reynolds number ($Re_{\delta} > 10^5$), several investigators have found that the separation length tends to become independent of the Reynolds number and even to slightly decrease with it, as indicated in Figure 29b. An explanation for this behaviour may be that at low Reynolds number the viscous sublayer represents a larger part of the total boundary layer and the viscous phenomena tend to dominate the interaction. At high Reynolds number, on the other hand, the viscous sublayer becomes exponentially thin. Therefore, interaction tends to be controlled by inertia and pressure forces (the influence of viscosity being minimised). Furthermore, at high Reynolds number, the subsonic layer is far thicker than the laminar sublayer. As a consequence of these two facts, pressure propagation in a high Reynolds boundary layer is essentially an inviscid mechanism.

![Figure 29. Influence of Reynolds number and ramp angle on separation length a) at low to moderate $Re_{\delta}$, $L_s/\delta$ increases with $Re$, data from Spaid and Frishett b) at high $Re_{\delta}$, $L_s/\delta$ decreases with $Re$, data from Settles.](image_url)

Besides the Reynolds number, heat transfer influences the separation length. The cooling effect can be found in Figure 30, where $L_s/\delta$ is plotted versus $T_w/T_r$ based on experimental data from Spaid and Frishett. $L_s/\delta$ is the ratio between $L_s/\delta$ when heat transfer is present and $L_s/\delta$ with adiabatic flow evaluated at the same $Re_{\delta}$. As indicated in the figure wall cooling decreases the separation distance. This reduction of $L_s/\delta$ with decreasing wall temperature can be explained with the help of the free interaction theory. When reducing $T_w/T_r$ ($T_r$ is the wall recovery temperature), the skin friction coefficient will increase and according to Eq 21 this provokes a decrease of $L_s$. Another interpretation of the reduction of $L_s/\delta$ is that an overall contraction of the interaction
domain is obtained due to a thinning of the subsonic layer, as the temperature level and thus the speed of sound near the wall becomes lower.

\[
\alpha, \text{ deg} \\
\bigcirc 19.67 \\
\bigtriangleup 17.25 \\
\square 13.70 \\
\bullet 9.81 \\
\bigtriangleup 8.07 \\
\blacksquare 7.52
\]

\[
\tilde{L}_s = \frac{(L_s/\delta)_{\text{Cooled wall}}}{(L_s/\delta)_{\text{Adiabatic wall at same } Re_\delta}}
\]

\[
T_w \\
T_r
\]

Figure 30. Influence of wall cooling on the separation length in a ramp flow. \( M_i=2.9 \), ramp angles \( 7.52^\circ \leq \alpha \leq 19.7^\circ \), \( 2.18 \times 10^4 \leq Re_\delta \leq 5.92 \times 10^4 \) and \( 0.474 \leq Tw/Tr \leq 1.05 \) (data from Spaid and Frishett\(^{54} \)).

5.2.4 Unsteadiness and 3-dimensional effects

In the previous section we only looked at the mean properties of shock induced separation. However, the shock-wave boundary layer interaction is an intrinsically unsteady phenomenon. This unsteadiness may generate large fluctuating forces that can be of severe magnitude e.g. in flight vehicles and overexpanded rocket nozzles, and has therefore been the topic for several studies\(^{55-56} \). A typical distribution of the fluctuating pressure \( p' \) in the interaction region is shown in Figure 31. The fluctuations increase rapidly after the onset of the interaction at \( I \) from the level experienced in the incoming unperturbed boundary layer, \( p'_i \), up to a peak value. It then decreases asymptotically towards the fluctuation level, \( p'_p \), in the plateau region.

The explanation of the obtained feature, first given by Kistler\(^{63} \), is that the flow is intermittent. In the interaction region the pressure jumps back and forth between the mean pressure levels \( p_i \) and \( p_p \) due to a fluctuation of the separation point, and at each pressure level the pressure oscillates with an amplitude characteristic of that level, i.e. \( p'_i \) and \( p'_p \) respectively, see Figure 32.

According to Kistler, the wall pressure signal near the separation can be modelled as a step function, with the jump location (i.e. the shock wave) moving over some restricted range. By defining \( \epsilon \) as the fraction of time that the plateau pressure region is acting over the point of interest, i.e. an “intermittence” factor, the mean pressure at a given axial position \( x \) can be expressed as:

\[
p(x) = \epsilon(x)p_p + [1 - \epsilon(x)]p_i
\]

Eq. 25

Thus, \( \epsilon \) can be determined from mean pressure measurement at \( x \) as:
The mean-square fluctuation around the mean pressure then becomes:

\[
\varepsilon(x) = \frac{p(x)/p_i - 1}{p_p/p_i - 1}
\]

Eq. 26

\[
p^{''2}(x) = \varepsilon[1 - \varepsilon'(p_p - p_i)]^2 + \varepsilon\rho_p^{''2} + [1 - \varepsilon']p_i^{''2}
\]

Eq. 27

In Figure 33 the results from such a calculation are compared to test data from Kistler, showing a good quantitative agreement between the measured and computed results.

Figure 31. Typical distribution of the fluctuating pressure in the interaction region.

Figure 32. Sketch of the time variation of the pressure within the interaction domain.
A better understanding of the instantaneous pressure profiles was obtained by Erengil and Dolling\textsuperscript{R61}. Their experiment showed how the time averaged wall pressure profile is composed of instantaneous profiles with sharper gradients see Figure 34. The figure shows the ensemble-averaged pressure $P_{\text{E/A}}$ in a Mach 5 compression ramp interaction. The profiles were obtained by picking out instants when the shock was located at various specific positions ($n=1..8$) in the intermittent region and ensemble-average them at each position. The solid black line in the figure represents the averaged mean pressure. The mean separation begins at $s=1$ and the flow reattaches somewhere downstream of the corner, which is located at $s=1.7$. As reported by Erengil and Dolling, three features of the ensemble averaged pressure distributions are evident:
1. For shock locations with $s > 1.2$, i.e. in the separated flow region, the shock position in the intermittent region has no significant influence on $\bar{p}_{E/A}$.

2. For the “shock-upstream” case, i.e. $n=1$, a well-defined plateau region can be seen in $\bar{p}_{E/A}$, consistent with a large-scale separated flow.

3. As the shock moves downstream from the $n=1$ position i.e. $s > 0.12$, a progressive change in $\bar{p}_{E/A}$ can be seen, to finally resemble that typical of a flow with a small separated region.

On the basis of fluctuating pressure measurements\textsuperscript{R 59}, laser field imaging methods\textsuperscript{R 71-R 73} and numerical simulations\textsuperscript{R 74} different researchers have suggested that the obtained shock motion is due to turbulent velocity fluctuations in the upstream boundary layer. A simple explanation given in reference R 71 is that changes of the shape of the instantaneous turbulent velocity profile yields changes in the shock position and hence produce the unsteady shock behaviour. With positive velocity fluctuations a fuller velocity profile is obtained which has a increased resistance to separation and the shock location will hence move further downstream. With negative velocity fluctuations the case will be the opposite. Since the velocity fluctuations are random they cause a random distortion of the separation line in the spanwise direction. This indicates that shock wave boundary layer interactions in nominally 2D flows are in fact a 3D phenomenon. This 3D effect is clearly seen in the surface streak pattern Settles\textsuperscript{R 42} obtained by applying a mixture of graphite powder and silicon oil to the ramp model surface, see Figure 35. A well-defined accumulated separation line can be seen in the figure upstream the corner. Close examination of the line shows that many variable length streaks project forward of it and these are records of instants when the reversed flow extended further upstream. Thus the instantaneous spanwise separation line is ragged, i.e. the flow is 3-dimensional, and the accumulated separation line represents the downstream boundary of a band of separation. This also explains why the flow picture close to the separation is blurred in instantaneous schlieren photos taken normal to the flow, as it gives an averaged picture of the flow in the spanwise direction.

Normalised power spectra for a variety of flow types are all broadband, with a large fraction of the energy at relatively low frequencies. In this context, low frequency means low relative to the characteristic frequency, $U_\infty/\delta_i$, of the incoming turbulent boundary layer. An example from Erengil and Dolling\textsuperscript{R 61} in Figure 36 shows power spectra in the interaction region in a 28°, Mach 5 compression ramp interaction. The ratio of $U_\infty/\delta$ for this flow is about 50 kHz, which can be seen in Figure 36 d). At $\varepsilon=0.06$ and $\varepsilon=0.80$, Figure 36 b)-c), the intermittence is low and the spectra is bimodal, reflecting the contributions from both the shock-induced fluctuations (about 0.2-2 kHz) and the undisturbed and separated boundary layer. Within the intermittent region, the power spectra retains the same shape with a large fraction (≈80-90%) of the energy below $f_{max}=2$ kHz, see definition in Figure 36 f). In the separated flow region the pressure fluctuations are caused by the turbulent activity in the free shear layer near the dividing streamline and a increased contribution from high frequencies is again evident, see Figure 36 a). This general trend is typical for all spectral results given in the literature. The power spectra at maximum rms, i.e. at $\varepsilon=0.5$, of different type of flows indicate frequencies of the broadband fluctuation below $f_{max}=1-10$ kHz\textsuperscript{R 58, R 61, R 63, R 64}. Analysis shows that the data are well correlated, when normalising the maximum frequency value with $L_s/U_\infty$, corresponding to Strouhal numbers $Sr = f_{max}L_s/U_\infty = 0.07$ for these configurations, see Table 2. This indicate, that with increasing separation length an increasing fraction of the energy will be located at low frequencies, i.e. closer to the eigenfrequencies of the structure. Why an increased forced response load can be expected.
Figure 35. Surface streak patterns in a 24°-ramp flow at $M_\infty = 2.85$, from Settles$^{R, 42}$, C-Corner, S-Separation, R-Reattachment. (Courtesy of Settles$^{R, 42}$)

Figure 36. Typical power spectra in the intermittent region (adopted from Erengil and Dolling$^{R, 61}$). a)-d) spectra at different streamwise locations, d) sketch of the streamwise evolution of the rms wall pressure and locations where the spectra have been evaluated, f) definition of $f_{\text{max}}$. 

$G(f) \cdot f / \sigma^2 p'_w$

$e = 1.00$

$e = 0.80$

$e = 0.06$

$e = 0.00$

$e = 0.50$

$f_{\text{max}}$

$f$ [kHz]
Table 2. Obtained Strouhal numbers for different flow configurations, when normalising the maximum frequency value with $L_s/\delta_i$.

In summary the shock wave boundary layer interaction is an intermittent and 3 dimensional phenomenon. Mechanical structures exposed to this type of supersonic flow separation are affected by large, time-dependent forces, which can be resolved into two components, a low frequency buffeting caused by changes in the geometry of the separation region, and high frequency fluctuations originating from the shear layer of the separated region.
A flow exposed to an adverse pressure gradient of sufficient strength can cause the boundary layer to separate from the wall. In the previous section we examined the influence of such adverse pressure gradients generated by obstacles. A similar condition occurs when a nozzle is operating in an overexpanded condition. A nozzle flow is said to be overexpanded when the theoretical wall exit pressure \( p_{e, \text{vac}} \) (the wall pressure obtained when the flow is ejected into vacuum ambient conditions) is below the ambient pressure \( p_a \). Thus at overexpanded flow condition the ratio \( n = p_{e, \text{vac}}/p_a < 1 \). \( n \) is a parameter commonly used to define the flow condition (adapted condition \( n=1 \) and underexpanded condition \( n>1 \)). As soon as \( n \) is slightly reduced below one, an oblique shock system is formed from the trailing edge of the nozzle wall due to the induced adverse pressure gradient. When the ratio \( n \) is further reduced, to about 0.4-0.8, the viscous layer cannot sustain the adverse gradient imposed upon it by the inviscid flow and the boundary layer separates from the wall. This is the case e.g. when a rocket engine designed for altitude operation is tested at sea level. It also occurs during start transients, shut off transients, or engine throttling modes. In order to provide scientists and engineers with information on the turbulent shock wave boundary layer interaction in overexpanded nozzles, many experiments have been carried out both in the past and recently for full scale and subscale nozzles, see e.g. reference R 5-R 8, R 81-R 89. Further support to the analysis of the flow separation behaviour has been provided by means of numerical simulationsR2, R4, R5, R90-R93, R104.

Recent research has made it clear that two different separation patterns have been observed, the classical free shock separation, and the restricted shock separation, in the following denoted by their acronyms FSS and RSS respectively. Figure 37 shows a schematic figure for both separations patterns with the definition of the characteristic points. In addition, Figure 38 compares measured and numerically calculated wall pressures for both separated flow patterns, and also includes the numerically calculated Mach number distribution for FSS and RSS, respectively.

### 6.1 FREE SHOCK SEPARATION

In the free shock separation case, the overexpanded nozzle flow fully separates from the wall at a certain ratio of wall- to ambient pressure. The resulting streamwise wall pressure evolution is mainly governed by the physics of shock wave boundary layer interactions occurring in any supersonic flow separation, see section 5.2. The first deviation of the wall pressure from the vacuum profile\(^1\) is commonly named *incipient separation pressure*, \( p_i \) in Figure 37 (\( p_i \) is some times also labelled \( p_{sep} \)). The wall pressure then quickly rises from \( p_i \) to a *plateau pressure* \( p_p \), which is in general slightly lower than the ambient pressure \( p_a \). Analyses of subscale tests in nozzles\(^1\) R 79-R 80, R 108 and flow with obstacles\(^1\) R 55-R 73 have shown that the steep pressure rise is caused by fluctuations of the shock front. The shock motion between the incipient separation point \( x_i \) and the point where the plateau pressure is reached, \( x_p \), is a low frequency wideband fluctuation with about 80-90% of the energy below the maximum frequency at \( Sr = f_{max} L_s/U_s \approx 0.07 \), see section 5.2.4. This observation, although not completely new, is in contrast to the classical view of a stable and well defined separation point, where the pressure rise from \( p_i \) to \( p_p \) has the origin in compression waves focussing to the oblique separation shock\(^1\) R 83, R 94.

From many cold gas tests in the past decades it has been noticed, that the boundary layer effectively separates from the nozzle wall shortly before reaching the plateau pressure \( p_p \). In the recirculation zone downstream of the separation point, the wall pressure increases slowly from \( p_p \) to \( p_e \), see Figure 37 (top). This gradual pressure rise is due to the inflow and upstream acceleration of gas from the ambience into the recirculation region.

To predict the axial separation location inside a nozzle, the ratio of separation to ambient pressure \( p_i/p_a \) must first be known. Using the vacuum wall pressure profile in the nozzle, the separation location can then easily be determined. Of course, the separation pressure ratio \( p_i/p_a \) includes the influence of both the pressure rise at the separation location itself and the gradual pressure rise in the recirculation region. To simplify the

\(^1\) The wall pressure profile obtained in the nozzle when the gas is expanded into vacuum ambient condition is called the vacuum pressure profile.
physical interpretation of the separation pressure ratio \( p_r/p_w \), it should be subdivided into two factors \( (p_r/p_p)(p_p/p_w) \), where each part refers to a single physical phenomenon, the former to the separation itself, the latter to the subsequent open recirculation with inflow of ambient gas.

Figure 37. Phenomenological sketch of free shock separation (FSS, top), and restricted shock separation (RSS, bottom).

It was noticed already in the early 1950’s, that the separation pressure ratio decreases during the start-up of nozzle flows, as the separation point moves downstream with increasing pressure ratio \( p_r/p_w \). This was soon attributed to the Mach number influence, as experiments in wind tunnels had shown the separation pressure ratio to decrease with increasing Mach number. However, there is a deviation from this regular behaviour as the separation point reaches the vicinity of the exit. At a location where the local area
ratio of the nozzle has reached about 80% of its final value, the separation pressure ratio, $p_i/p_a$, reverses its previous trend and increases as the pressure ratio $p_i/p_a$ is increased. An explanation given for this behaviour in reference R 88 is that close to the nozzle exit the plateau pressure increases to ambient pressure. For a constant pressure ratio, $p_i/p_a$, this would cause an effective increase in separation pressure, $p_i$, in this last part of the nozzle, and thus an increase in $p_i/p_a$. As the pressure plateau $p$ reaches the nozzle exit, the flow is actually attached all the way to the exit even though the sensors detect a clear pressure rise. This is usually referred to as incipient separation at the nozzle exit.

Figure 38. Free (left) and restricted shock separation (right) in the parabolic subscale nozzle VOLVO S1, comparison of measured and calculated wall pressures, and calculated Mach number distribution. Experimental data by FOI calculations performed by VOLVO (from Östlund R5).

6.2 RESTRICTED SHOCK SEPARATION

During cold-flow subscale tests for the J-2S engine development in the early 70s, a different kind of separated nozzle flow was observed at strongly overexpanded conditions, which had not been known before. R 95 In this flow regime, which only occurred at certain pressure ratios, the pressure downstream of the separation point showed an irregular behaviour and partly reached values above the ambient pressure. This is attributed to a reattachment of the separated flow to the nozzle wall, inducing a pattern of alternating shocks and expansion waves along the wall. Due to the short separated region, this flow regime was called restricted shock separation. The separation characteristic of restricted shock separation, as observed in the literature R 95, and recently confirmed for subscale R 5, 7, 8 and full-scale rocket nozzles R 2 - 4 is described in the following.

During the start-up of the nozzle flow, featuring initially pure free shock separation, the transition from FSS to RSS occurs at a well-defined pressure ratio. R 4, R 5 A closed recirculation zone is formed, with static pressures significantly below the ambient pressure level. Thus, the transition from FSS to RSS is connected with a sudden downstream movement of the separation point. Beyond the reattachment point in RSS, supersonic flow propagates along the nozzle, thereby inducing shocks that result in the aforementioned wall pressure peaks above ambient pressure. By further increasing the thrust chamber pressure ratio, the closed recirculation zone is pushed towards the nozzle exit. Finally, the reattachment point reaches the nozzle exit, and the recirculation zone opens to the ambient flow. This is connected with a pressure increase in the recirculation zone behind the separation shock, which pushes the separation point again further upstream. Thereby it occurs, that the recirculation zone closes again, connected with a drop in static pressure, which
results again in a downstream movement of the separation point. A pulsating process is observed, connected with the opening and closing of the separation zone. This re-transition from RSS back to FSS is in the literature also referred to as the end effect.\cite{R4,R5}

The same phenomena can be also observed during shut-down. While the end-effect, and thus the transition, now from FSS to RSS, occurs at the same pressure ratio as the RSS to FSS transition during start-up, the re-transition from RSS to FSS occurs in general at a different lower pressure ratio than the corresponding transition FSS-RSS during start-up.\cite{R2,R5}

The theory of reattached flow in the J-2S sub-scale nozzle was first confirmed by numerical simulations of Chen et al. in 1994\cite{R90}. In addition, their calculations revealed a trapped vortex behind the central normal shock, but they did not provide any explanation for the generation of such flow structure. Later, Nasuti and Onofri\cite{R91,R93} stressed the role played by the centreline vortex on the separation pattern and side-load generation. The centreline vortex acts as an obstruction for the exhausting jet, which is thereby pushed towards the wall. As a consequence a radial flow component is generated that tends to reattach the separated region, thus switching the flow from FSS to RSS.

Frey and Hagemann have given another explanation of the reattached flow based upon experimental observations and numerical simulation.\cite{R2,R3} According to their results, the key driver for the transition from FSS to RSS and vice versa is the specific cap-shock pattern. Thus, a transition from FSS to RSS can only occur in nozzles featuring an internal shock. According to their findings, the cap-shock pattern results from the interference of the separation shock with the inverse Mach reflection of the weak internal shock at the centreline.\cite{R3} A key feature of this inverse Mach reflection is the trapped vortex downstream of it, driven by the curved shock structure upstream of it which generates a certain vorticity in the flow.\cite{R3,R96,R97} Thus, the vortex would be a result of the curved shock structure, which is partially in contrast the explanation given by Nasuti and Onofri, that includes also an effect of flow gradients upstream. Further experimental and numerical verification is planned to finally conclude on the interesting vortex phenomenon.

However, it is interesting to note that both hypotheses of Nasuti and Onofri, and Frey and Hagemann identify the curved cap-shock profile as driver for the transition from FSS to RSS, which is meanwhile proven by experiments.\cite{R4,R8}

6.3 CRITERA FOR FLOW SEPARATION PREDICTION IN ROCKET NOZZLES

6.3.1 Free shock separation criteria

The theoretical prediction of free shock separation is the case, which has been most extensively studied in the past since, historically, almost all experiments have been performed in conical and truncated ideal nozzle contours only featuring this separation pattern. Experimental data have been used to develop a number of empirical and semi-empirical criteria in order to give the nozzle designer a prediction tool for the separation point, although knowing that in reality there is no exact point of separation because it fluctuates between two extreme locations. But even today, an exact prediction cannot be guaranteed because of the wide spectrum of parameters involved in the boundary layer – shock interaction such as nozzle contour, gas properties, wall temperature, wall configuration and roughness.

Probably the most classical and simple criteria for FSS purely derived from nozzle testing is the one given by Summerfield et al.\cite{R83} which is based on extensive studies on the separation phenomenon in conical nozzles in the late 1940’s:

\[ p/p_a = 0.4 \]  

Eq. 28

A first approach to include the Mach number influence was published by Arens and Spiegler in the early 1960’s.\cite{R87} However, the major formula derived turned out to be too complex for engineering application. Based on experiments with conical and truncated ideal nozzles, Schilling derived in 1962 a simple expression accounting for the increase of separation pressure ratio \( p/p_a \) with increasing Mach number,
\[
\frac{p}{p_a} = k_1 \left( \frac{p_r}{p_a} \right)^{k_2}
\]

Eq. 29

with \(k_1 = 0.582\), and \(k_2 = -0.195\) for contoured nozzles, and \(k_1 = 0.541\), and \(k_2 = -0.136\) for conical nozzles.\(^{R85}\)

In 1965, based on Schilling’s expression Kalt and Badal chose \(k_1 = 2/3\) and \(k_2 = -0.2\) for a better agreement with their experimental results.\(^{R86}\) NASA adopted a correlation similar to the one of Schilling for truncated contoured nozzles as a state of the art indication at the mid 1970’s.\(^{R98}\)

Later investigations performed by Schmucker\(^{R94}\) lead NASA to recommend the semi-empirical criterion by Crocco and Probstein\(^{R99}\), which is based on a simplified boundary layer integral approach. The criterion accounts for the properties of the boundary layer, the gas and the inviscid Mach number at the onset of separation. The NASA recommendation from 1976 was to use this criterion with an additional margin of 20% from the predicted separation occurrence. Another inheritance from this time is the purely empirical criterion proposed by Schmucker: \(^{R94}\)

\[
\frac{p}{p_a} = (1.88M_i - 1)^{-0.64}
\]

Eq. 30

which has similar characteristics as the Crocco and Probstein criterion and is still widely used.

Figure 39. Comparison of simple separation prediction models for \(p/p_a\) with experimental results. The symbol shape in the legend indicates from which investigation the data is taken and the symbol colours correspond to different nozzle configurations tested, see Frey.\(^{R110}\) Also published in R 105. (remark \(p_{sep}=p_i\) and \(M_{sep}=M_i\))

In Figure 39 a comparison between these criteria with test data are shown. As indicated in the figure a significant scatter of the data points can be observed. This explains the NASA advice of a 20% margin and also points out the necessity of new and more reliable criteria. One of the major reasons for the rather poor agreement is that all above criteria include two separate mechanisms involved in the pressure rise of the flow in one single expression. This fact was realised already in the 1960’s by Arens and Spiegler\(^{R87}\), Carriere\(^{R100}\) and by Lawrence.\(^{R89}\) The latter suggested that the pressure recovery \(p/p_a\) should be subdivided into two parts, one part for the critical pressure rise, \(p/p_r\), over the separation shock and a second for the pressure rise in the recirculation zone, \(p/p_a\).

The pressure rise \(p_r\) to \(p_p\) is caused by shock-wave boundary layer interaction, as described in paragraph 5.2. This is a general mechanism, not restricted to nozzle flow separation, which has been extensively studied. As
an example, Zukoski [R45] found the following simple relation (cf. Eq. 24) to be in good agreement with experimental results for high Reynolds numbers:

\[ \frac{p_i}{p_p} = \left(1 + 0.5 M_i \right)^{-1} \]  

Eq. 31

for the Mach number range of \( M_i = 1.4-6.0 \) and \( Re_\alpha > 10^7 \). According to the author, this correlation also agrees with the plateau pressure values measured in overexpanded conical nozzles in the Mach number range \( M_i = 2.0-5.5 \).

The drawback of the Zukoski criterion is that it does not include the dependency of the specific heat ratio observed in experimental data and should thus only be used for gas flow with \( \gamma = 1.4 \), since the experiments were performed with air. A first attempt to account for the specific heat ratio dependency by using oblique shock relations was proposed by Summerfield et al. 1954. [R83] From experimental data they found that the flow deflection angle \( \theta \) of the separated flow was nearly constant \( \theta = 15^\circ \) for the nozzles tested. With this value and the use of oblique shock theory the pressure rise for different gas mixtures can thus be calculated. This observation has also been confirmed in later synthesis of nozzle flow separation data, from a number of experiments performed with both hot and cold gas flows [R2]. However, the data also indicate that the Summerfield criterion with a constant \( \theta \) value is too simple. In fact the data rather indicate a linear dependence of the Mach number on both the deflection angle \( \theta \) and the shock angle \( \beta \) itself. Based on this and data from the VOLVO subscale tests [R6] Östlund [R109] proposed an empirical criterion based on oblique shock relations:

\[ \frac{p_i}{p_p} = \left(1 + \gamma M_i^2 \sin^2 (\beta) \left[ 1 - \frac{\tan(\beta - \theta)}{\tan(\beta)} \right] \right)^{-1} \]  

Eq. 32

with \( \beta = -3.764 M_i + 42.878 \) [°] and \( \theta = 1.678 M_i + 9.347 \) [°] for the Mach number range 2.5 ≤ \( M_i \) ≤ 4.5. Östlund used linear expressions for both \( \theta \) and \( \beta \) in the correlation since he found that a criterion only based on the shock angle \( \beta \) (and \( \theta \) calculated with the \( \theta - \beta - M \) relation) experiences a minimum already for a modest extrapolation above \( M_i = 4.5 \). Frey [R110] has proposed a similar criterion based only on the shock angle \( \beta \) as:

\[ \frac{p_i}{p_p} = \left[1 + \frac{2\gamma}{\gamma + 1} \left( M_i^2 \sin^2 (\beta) - 1 \right) \right]^{-1} \]  

Eq. 33

with \( \beta = 4.7 M_i + 44.5 \) [°] for the Mach number range 2.5 ≤ \( M_i \) ≤ 4.5, which produces a similar result as the criterion by Östlund (Eq. 32 reduces to Eq. 33 with the use of the \( \theta - \beta - M \) relation). However, it does not give the correct trend of \( p_i/p_p \) for higher Mach numbers. At \( M_i = 4.8 \) the function has a minimum and \( p_i/p_p \) suddenly increases with the Mach number.

Although these criteria give a significant improvement, they are still pure empirical and it is always better to base a criterion on a physical model in order to correctly include the influence of governing parameters. A promising theory to build such a criterion on seems to be the generalised free interaction theory by Carrière et al. [R111], which has received new attention within the FSCD group [R159, R160]. These authors generalised the free interaction theory by Chapman, see 5.2.2, by taking into account both non-uniformity in the incoming outer flow and the wall curvature in the interaction region. They found that in the most generalised case, the universal correlation function takes the form (cf. Eq. 19-20):

\[ F\left( \frac{x-x_r}{x_r-x_i}, p' \right) = \frac{p(x)-p_i}{q_i} \frac{\bar{V}(x)-V(x)}{C_{\bar{f}}} \] , \( p' = \frac{\delta^*}{q_i} dp/dx \)  

Eq. 34

Where \( V \) is the Prandtl-Meyer function for the actual pressure at \( x \) and \( \bar{V} \) the value \( V \) would take at the same location in absence of flow separation. \( p' \) is the normalised pressure gradient characterising the non-uniformity of the flow. For a specific \( p' \), the function \( F \) is assumed to be a universal correlation function.
independent of Mach and Reynolds numbers, to be determined from experiments. In Figure 40 the generalised wall pressure correlation function for non-uniform flow, \( F \), and the separation length, \( l_s \), obtained by Carrière et al. is shown. The correlation function for uniform flow is also included in the figure so the influence of \( p' \) on \( F \) can be seen. Carrière et al. based their correlation on axi-symmetrical experimental data from one ideal nozzle with design Mach number \( M_D = 3 \) and three conical nozzles with half-angles of 5º, 10º and 17.5º respectively. With these nozzles the Mach number range \( 2.06 \leq M_i \leq 2.78 \) and \( 4.12 \leq M_i \leq 5.04 \) was covered for values of the pressure gradient in the range \(-1.2 \leq p' \times 10^3 \leq -0.8\).

Figure 40. Free interaction theory. Pressure correlation and separation length for non-uniform flow, \( F = 4.22 \), Carrière et al.\(^{111}\).

However, in order to obtain a criterion for the pressure rise \( p_i/p_p \), a correlation for the interaction length \( l_p \), i.e. from the start of the shock boundary layer interaction to the plateau point, is needed rather than the separation length \( l_s \) itself as given by Carrière et al. Since no rational definition of the plateau point exists for non-uniform flow, we define the plateau point to be the position where the function \( F \) has the value \( F_p = 6.0 \) characterising the plateau point in uniform flow. A first attempt to calculate the pressure correlation function \( F(s, p') \) and the interaction length by only exploiting data from the VOLVO subscale test campaigns\(^{R6} \) showed to be unsuccessful due to insufficient axial resolution of the pressure measurements in the interaction region. Instead the pressure correlation function by Carrière et al. was fitted to the experimental data by varying the values of \( x_i \) and \( l_s \) for each nozzle flow condition. Results from such a procedure, applied to experimental data obtained with the VOLVO S6 nozzle, are shown in Figure 41. The flow properties were determined using TDK together with the built in boundary layer module for each operational condition. As can be seen in the figure, the pressure correlation curve by Carrière et al. fits the experimental data well. The corresponding calculated values of the separation length and the plateau length are shown in Figure 42. The obtained values of the separation length and the ones given by Carrière et al. are very similar. The small difference is probably due to differences in the computational method used for determining of the boundary layer properties. A correlation function for the plateau length i.e. \( l_p / \delta_* \) was then determined with the use of a least square curve fitting technique. The obtained correlation curve is indicated in Figure 42. In contrast to the free interaction theory for uniform flow, see 5.2.2, the interaction length for the non-uniform flow condition obtained in overexpanded nozzles also depends on the downstream conditions. The influence of e.g. the plateau pressure value on interaction length can be found by rewriting equation 34 at the plateau point as:

\[
\frac{F_p}{v_p - v_p} = \frac{p_i / p_p - 1}{\frac{1}{2} \gamma M_i^2 C_p F_p}
\]

Eq. 35
Inspection of equation 35 together with Figure 42 shows that \( l_p/\delta^* \) increases as the plateau pressure is reduced, which has also been verified in experiments\(^{111} \). However, other empirical relations used for the interaction length in separated nozzle flows show no influence of the downstream conditions, e.g. Dumnov et al.\(^{112} \) found that \( l_r/\theta^*=f(M_i,T_w,i) \), where \( \theta^* \) is the momentum thickness at the start of the interaction.

Figure 41. Fit of generalised pressure correlation curve by Carrière et al. to VOLVO S6 data, \( \chi_i \) and \( l_s \) varied, \( 2.82 \leq M_i \leq 3.25, -0.9 \leq p' \times 10^3 \leq -0.5, n=0.04-0.24. \) (Data also published in Östlund et al.\(^{108} \))

Figure 42. Interaction length correlation, to separation point (\( l_s \)) and plateau point (\( l_p \)) respectively. Symbols indicate calculated values based on VOLVO S6 nozzle test data.
With the use of equation 35 and the correlation function for the plateau length, the location of the start of the interaction process \((x_i)\) can be determined in a nozzle at a given operation condition when the plateau pressure value is known. This is done by iteratively solving the implicit equation 36.

\[
\frac{F_p}{\nabla(x_i + \frac{x}{\delta^*}(f_i) \cdot \delta^*(x_i)) - V_p(p_p)} = f_i, \tag{Eq. 36}
\]

with \(f_i(x_i, p_p) = \frac{p_p}{\frac{1}{2} \gamma M_i^2 C_f F_p} \). 

In order to check the validity of this criterion it was applied to the Volvo S7 short nozzle. The flow properties at different operational condition were determined using TDK together with the built-in boundary layer module BLM. For each flow condition a plateau pressure value was specified based on experimental data. As can be seen in Figure 43, the predicted pressure profiles in the interaction zone show a good agreement with the test data for all cases.

![Figure 43. Predicted and measured wall pressure profile in the Volvo S7 Short Nozzle.](image)

Although the first results look promising, a lot of effort needs to be put down before a reliable and accurate criterion can be established. More test data need to be evaluated in order to increase the accuracy of the correlation functions and the applicability to chemical reacting flow cases, where the value of the specific heat ratio is different compared with air, must also be validated. The influence of wall cooling has to be examined, especially the wall temperature effect on the interaction length. One simple method to take this influence into account can be to formulate a correction function \(\frac{l_{r, \text{cooled}}}{l_{r, \text{adiabatic}}} = f(M_i, T_w/T_r)\) similar to the approach used by Lewis et al. for laminar flow\(^{113}\), see also the results obtained by Spaid and Frishett\(^{84}\) for turbulent ramp flow in Figure 30. The scaling of the interaction length with the displacement thickness, \(\delta^*\), must also be revised since \(\delta^*\) can become negative in strongly cooled nozzle flows. The boundary layer thickness, \(\delta\), or the momentum thickness, \(\theta\), may be a better choice for scaling in such cases. In order to shed light on these opened ends, test are presently being prepared at Volvo, ASTRIUM and DLR with some test objectives specially focused on the wall temperature effects on nozzle flow separation. An example taken from the preparations of this campaign is shown in Figure 44.
As shown above, the streamwise length of the interaction zone cannot be predicted with the generalised free interaction theory alone since it depends on the flow in the separated region. It needs to be coupled with a model describing the flow downstream of the shock-wave boundary layer interaction, where the pressure recovery, $p_p/p_a$, takes place. Such a model is currently not available for contoured nozzles. The only reported models for the recirculating flow in the literature are the ones by Kudryavtsev\(^1\) and the one by Malik and Tagirov\(^2\), both for conical nozzles operated with air. The model by Kudryavtsev is purely empirical. He found that in conical nozzles with a half angle $\alpha < 15^\circ$ the pressure rise in the recirculating zone could be approximated as:

$$p_p/p_a = \left[1 + \frac{0.192}{\sin \alpha} - 0.7 \left(1 - \frac{M_a}{M_s}\right)^2\right]^{-1}$$

Eq. 37

Where $M_s$ is the average exit Mach number defined by the nozzle expansion area ratio $\varepsilon$. Whereas, in conical nozzles with a half angle $\alpha > 15^\circ$ he found that the pressure rise $p_p/p_a=1$, i.e. independent of the Mach number. The pressure rise calculated with equation 37 is shown in Figure 45 for conical nozzles with half angles $5^\circ \leq \alpha \leq 15^\circ$.

The model by Malik and Tagirov on the other hand is semi-empirical and is based on Abramovich’s theory for the mixing of counterflowing turbulent jets.\(^1\) This model shows good agreement with test data and if it is generalised it could be a promising model for contoured nozzles operated with hot propellants. A model for recirculating flow in contoured nozzles, whether empirical or semi-empirical, must take into account a number of parameters. Experimental data indicate e.g. that the wall contour downstream the separation point has a significant influence on the pressure increase in the recirculation zone.\(^2\) As reported in reference R 2, the length of the separated region, the curvature of the wall downstream of the separation and the radial size of the recirculating zone between the wall and the jet are further parameters influencing the pressure rise $p_p/p_a$. A clear indication of this can be found in Figure 46, where $p_p/p_a$ is plotted versus $\varepsilon - \varepsilon_i$, which is a measure of the radial size of the recirculation zone. For large values of $\varepsilon - \varepsilon_i$, the downstream contour has a negligible influence on the pressure rise, whereas for the case when the separated jet is close to the wall (small $\varepsilon - \varepsilon_i$) there is a large variation in $p_p/p_a$. Besides that, the sudden increase of $p_p/p_a$ as the incipient separation point enters the nozzle exit region must also be included. This increase of $p_p/p_a$ is a general feature
for all nozzle flows and it is illustrated in Figure 47 by experimental values obtained with the short VOLVO S7 nozzle.

Figure 45. Pressure rise in the recirculating zone in conical nozzles with half angles $\alpha<15^\circ$ according to the model by Kudryavtsev.

Figure 46. Experimental results for the pressure rise $p_r/p_a$ as function of separation location. The symbol shape in the legend indicates from which investigation the data is taken and the symbol colours correspond to different nozzle configurations tested, see Frey. Also published in R 105.
Thus, it is obvious that in order to predict the location of separation successfully, a separation criterion must consist of two parts, first of all a model where the shock-boundary layer interaction is properly described and secondly a model where the pressure rise in the recirculating zone is included which accounts for downstream conditions and nozzle geometry. The development and validation work of such models is currently ongoing at the different partners of the FSCD group, see e.g. the recent work by Reijasse and Birkemeyer\textsuperscript{R160}.

### 6.3.2 Restricted shock separation criteria

The prediction of restricted shock separation has only been addressed in the last years, see reference R 3 and R 6. The key point for the prediction of RSS is to predict the location where the transition from FSS to RSS takes place. The driving force for reattachment of the flow is when the radial momentum of the separated jet is directed towards the wall, which can occur with a cap-shock pattern. Whereas no reattachment is possible if the momentum is directed towards the centre-line, which is always the case with a Mach disc. Thus, by quantifying the momentum balance of the jet, the transition point can be determined. On this basis Östlund and Biger\textsuperscript{R6} proposed a simple empirical criterion for the prediction of transition from FSS to RSS, which relates the FSS-RSS transition to the axial position where the small normal shock at the centre-line coincide with the RSS separation front, see Figure 48. As indicated in Figure 48 and Table 3 this model shows very good results considering its simplicity.

| Case      | \( p_c \) for transition: \(
\frac{\text{predicted}}{\text{actual}}\) |
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Vulcain NE</td>
<td>1.05</td>
</tr>
<tr>
<td>VolvoS1</td>
<td>0.94</td>
</tr>
<tr>
<td>VolvoS3</td>
<td>1.00</td>
</tr>
</tbody>
</table>

Table 3. VOLVO-model for predicting at which chamber pressure, \( p_c \), there is an FSS to RSS transition, also published in Östlund et al\textsuperscript{R108}.
Frey and Hagemann have developed a more sophisticated and physical model. In this model, the FSS shock system is always prevailing before a possible reattachment is defined. Based on numerical flow field data, the cap shock pattern is re-calculated by a shock-fitting technique. By calculating the momentum balance across the cap shock pattern and the corresponding direction of the jet downstream of the cap shock pattern, the driving force for reattachment is evaluated and the location where the transition takes place is determined.

Both models account for the sudden pressure drop of the plateau pressure and the subsequent jump of the separation point when the flow reattaches and the separated region becomes enclosed by supersonic flow. Due to the complexity of the flow downstream of the reattachment point, which is characterised by subsequent compression and expansion waves, no models for this pressure recovery process exist so far. Instead, a constant value of the plateau pressure based on test data experience is often used. This value is kept until the RSS is transformed back into FSS and FSS criteria are applicable again. This transformation occurs either when the cap-shock is converted into the Mach disc or when the enclosed separation zone is opened up at the nozzle exit, as shown in Figure 48.

Based on numerical simulations of the cap shock pattern with the trapped vortex, Reijasse has proposed a further transition prediction model based on an effective area ratio for the RSS condition, estimated with the effective nozzle exit area occupied by the re-attached annular jet, and the throat area. Thus, the remaining exit area filled with the re-circulating flow of the trapped vortex is ignored in this approach.
7 MEASUREMENT OF FLOW SEPARATION AND SIDE LOADS

7.1 STATIC WALL PRESSURE MEASUREMENTS

Static pressure taps are standard instrumentation in most flow applications and can give valuable information of the flow process if the instrumentation is carefully done and correct interpretations of the measurements are made. Figure 49 shows a typical wall pressure profile for a highly overexpanded nozzle flow. The location where the static wall pressure starts to rise, $x_s$, is the origin of the shock wave boundary layer interaction, and a correct determination of this position is essential for constructing flow separation models. In order to experimentally locate this position, an extremely narrow spacing of the pressure transducers is required. The separation length, $l_s = x_s - x_i$ (cf. Figure 25 and Figure 37), where most part of the pressure rise takes places ranges from 1 to 100 $\delta^*$, depending on operational condition, degree of overexpansion, nozzle contour etc., see Figure 42. A rough estimate, which should only be considered as a rule of thumb to get the order of magnitude, is $l_s < 0.5 r_t$, based on “cold” sub-scale test data see e.g. Figure 41 and Figure 43. As indicated in the schematic in Figure 49, a modest increase in the wall pressure takes place before a steep and almost linear pressure rise. To resolve this initial gradual increase and locate the first deviation from the vacuum pressure profile would require a transducer spacing of about $\Delta x = l_s/10 = 0.05 r_t$. In the VOLVO sub scale nozzles, which are considered to be of large scale in the context of cold flow model nozzles, the throat radius is $r_t = 33.54$ mm. Thus the order of magnitude of the separation length in these nozzles are about $l_s < 0.5 * 33.54 = 17$ mm, and to resolve the first deviation from the vacuum pressure profile would require a transducer spacings of 0.5 mm have been used at FOI R 137, however the instrumentation cost will become high and the transducers may influence the flow. The use of modern measuring techniques such as Pressure Sensitive Paint (PSP) would overcome this problem of spatial resolution. PSP is commonly used for external high-speed flow situations R 125, whereas there are only a few studies in the literature, where PSP has been applied to internal supersonic flow, see e.g. reference R 126-R 128. The main problem with PSP is that the paint has a strong temperature dependency. It must therefore be used together with IR-techniques or Temperature Sensitive Paint (TSP). In the case of internal flow, the visual access further complicates the situation. PSP is not as accurate as static pressure taps, the accuracy being only about 5%, why a combination of PSP and regular pressure taps is often used R 137.

Despite obvious practical problems to obtain the necessary resolution with static pressure taps, some investigators have adopted this approach to determine the origin of the interaction e.g. Carrière et al R 111. However, most investigators have chosen a conventional origin obtained by extrapolating to the wall the quasi-linear pressure rise at separation, as shown in Figure 49. The origin obtained with this method is always downstream of the physical origin. Another point of interest is the location of the end of the shock boundary layer interaction, i.e. the plateau point. In the case of obstacle-induced separation in uniform flow, a well defined plateau point can be observed in the wall pressure data, see Figure 25, however this is not the case in overexpanded nozzle flows as sketched in Figure 49. A common approach is to define the plateau pressure as the pressure value at the intersection between two straight lines, one line being tangent to the steep pressure rise obtained in the interaction region, and the other one tangent to the pressure rise in the recirculating flow region. Since the determination of the plateau point is rather arbitrary with this method, Östlund proposes inhere, to determine $x_i$ and $l_s$ from a best fit of the generalised pressure correlation function, $F(x_i, l_s)$ by Carrière et al R 111 to the experimental data. Since no rational definition of the plateau point exists for non-uniform flow, the plateau point is defined as the position where the function $F$ has the value $F_p = 6.0$ which is analogous to Erdos and Pallone R 78 definition of the plateau point in uniform flow, cf. Figure 27. The result from such a procedure, applied to experimental data obtained with the VOLVO S6 nozzle, is shown in Figure 41.
7.2 FLUCTUATING WALL PRESSURE MEASUREMENTS

As reported previous (see section 5.2.4 and 8.3), the shock boundary layer interaction found in highly overexpanded nozzle flows is an intermittent and three-dimensional phenomenon. Mechanical structures exposed to this type of supersonic flow separation are effected by large, time-dependent forces, which can be resolved into two components, a low frequency buffeting caused by changes in the geometry of the separation region, and fluctuations originating from the shear-layer of the separated region. Accurate models for the prediction of these loads are needed in order to mechanically define the thrust chamber structure to ensure mechanical integrity under “worst case” condition. Development and validation of such models still relies on generalisation of experimentally determined wall pressure fluctuations and their inter-correlation. Hence, the spatial resolution and accuracy of the fluctuating pressure measurements will determine the accuracy of the developed model. In practice however, the spatial resolution is limited and the sensors are placed where they can capture the most important events. Major part of the wall pressure load in a overexpanded nozzle featuring free shock separation originates from the pressure fluctuations in the shock boundary layer interaction region. In order to resolve the streamwise distribution of the rms pressure fluctuations in this region an extremely narrow spacing of the pressure transducers is required, as indicated in Figure 73-Figure 74. Based on Figure 73 we can see that an array of at least 5-10 pressure sensors along the interaction region would be needed in order to resolve the pressure rms distribution. The most important points to capture in this region are the point of maximum pressure rms and the locations of the origin and the end of the shock wave boundary layer interaction. The simplest way to capture the peak value is to find the operation condition when the peak is locked on a pressure transducer. This operation condition can be found by changing the pressure ratio $p_r/p_a$ with small stepwise increments or with a slow continuous ramp, as in Figure 74.
Since the gradients in the separated region are small, a fine streamwise resolution is not as important in this region as in the interaction region. Thus, 5-10 evenly spaced pressure sensors would be sufficient to capture the most significant features of the recirculating flow zone.

In order to quantify the instantaneous asymmetry of the pressure load, pressure sensors must also be installed in the transversal direction in the separation and the separated zone respectively. It is difficult to specify a minimum number of pressure sensors required in the transversal direction. In general reliable quantitative data on the structures and pressure fluctuations in the transverse direction are lacking and is fruitful area of future work. However, as an indication Dumnov used 8 fast pressure transducers in the transversal direction in order to obtain the pressure correlation function, on which he based his side-load model. For a more general survey of fluctuating wall pressure measurements for this type of flow, the work by Dolling and Dussauge is recommended, where method of measurement, common sources of error and calibration methods are discussed.

The development of fast pressure sensitive paint (FPSP) has evolved rapidly in recent years, see e.g. reference. FPSP have response times in the range 3-12 kHz. It may hence be possible in the near future to resolve the global unsteady pressure field in separated nozzle flows with this method.

7.3 SIDE LOAD MEASUREMENTS

Direct measurement of the global asymmetric fluctuating pressure load obtained during nozzle operation with flow separation would require a fast and global surface field measuring method, i.e. either the use of an enormous number of fast pressure transducers or the use of fast pressure sensitive paint. The commonly used measuring technique is to measure the mechanical side-load response caused by the aerodynamic side-load as it acts on the structure. If the structural dynamic transfer function is known the aerodynamic side-load can then be calculated. In a rocket engine the aerodynamic side-load can excite two different modes of the rocket engine structure. These modes are 1) the pendulum mode where the nozzle oscillates around the cardan and 2) the bending mode where the nozzle oscillates around the throat. An experimental set-up must simulate the most significant of these modes. In Figure 50-Figure 53 the experimental test set-ups used by VOLVO and ONERA are shown respectively (the test set-ups used at Keldysh and NAL are similar to the one used at DLR). The VOLVO test set-up simulates the bending mode whereas the experimental set-up at ONERA and DLR simulates the pendulum mode.

The device for measuring dynamic unsteady side-loads at ONERA consists of a support-tube equipped with semi-conductor strain-gauges in order to measure the bending moment in two planes, i.e. the two perpendicular components of the general side load moment. The reference point for these torque measurements is labelled CRB Tube in Figure 52.

The side-load measuring system in the DLR test facility P6.2 consists of one thin walled Aluminium pipe which connects the rigid test nozzle to the rigid gas feeding system, see Figure 53. As side-loads are produced in the nozzle, the thin walled pipe will bend and with the use of strain gauges the two perpendicular components of the resulting side-load response is measured. By changing the length of the strain-measuring pipe different system eigenfrequencies can be obtained.

In the test set-up used by VOLVO, the nozzle consist mainly of two parts, one fixed part mounted to the downstream flange of the wind tunnel and one flexible hinged part, see Figure 50. The flexible part is suspended with a universal joint/cardan, permitting motion in all directions around the throat and the motion simulates the throat-bending mode of a real rocket nozzle. The bending resistance is simulated with torsion springs, which are exchangeable, that the influence of the structure stiffness on the side load response and aeroelastic coupling can be studied. The side-load response components are measured with strain gauges mounted on the torsion springs.
Figure 50. Schematic side view of the experimental test set-up in FFA wind tunnel HYP 500, from Östlund et al.\cite{5,R6}.

Figure 51. Schematic side view of the cardan hinged test nozzle in FFA wind tunnel HYP 500, from Östlund et al.\cite{5,R6}.
Figure 52. Sketch of the experimental set-up in the ONERA R2Ch blowdown wind tunnel, from Reijasse et al.\textsuperscript{132}

Figure 53. Experimental test set-up and principles of the side-load measuring system in the DLR test facility P6.2, from Frey\textsuperscript{R110}
Since only the side-load response is measured in the test set-ups described above, a method for calculating the aerodynamic side-load is needed. During a run, the strain-gauges measure the strains, \( \tilde{S}(t) \), resulting from the dynamical response of the system to the aerodynamic side-load torque \( \tilde{M}_a(t) \). Where \( \tilde{S}(t) \) and \( \tilde{M}_a(t) \) each have two components representing motion in two directions around the main axis i.e.:

\[
\tilde{S}(t) = \begin{bmatrix} S_1(t) \\ S_2(t) \end{bmatrix} \quad \text{and} \quad \tilde{M}_a(t) = \begin{bmatrix} M_{a1}(t) \\ M_{a2}(t) \end{bmatrix}
\]

These are usually different due to the asymmetry introduced by the instrumentation and the test set-up.

After Fourier transform of \( \tilde{S}(t) \) and \( \tilde{M}_a(t) \) and introducing the transfer function \( H(f) \) we get:

\[
\hat{\tilde{S}}(f) = H(f) \hat{\tilde{M}}_a(f)
\]

Where \( H(f) \) is a complex 2x2 matrix defined by:

\[
H(f) = \begin{bmatrix} H_{11}(f) & H_{12}(f) \\ H_{21}(f) & H_{22}(f) \end{bmatrix} \quad \text{with} \quad H_{jk}(f) = |H_{jk}| e^{i\phi_{jk}}
\]

Once \( H(f) \) is known, the aerodynamic load components \( \tilde{M}_{a1}(t) \) and \( \tilde{M}_{a2}(t) \) can be reconstructed from the strain-gauge signals \( S_1(t) \) and \( S_2(t) \) as:

\[
\begin{bmatrix} \hat{\tilde{M}}_{a1}(f) \\ \hat{\tilde{M}}_{a2}(f) \end{bmatrix} = \begin{bmatrix} H_{11}(f) & H_{12}(f) \\ H_{21}(f) & H_{22}(f) \end{bmatrix}^{-1} \begin{bmatrix} \hat{\tilde{S}}_1(f) \\ \hat{\tilde{S}}_2(f) \end{bmatrix}
\]

### 7.3.1 Determination of the system frequency response function

As indicated above, the system frequency response function must be known in order to be able to reconstruct the aerodynamic load components. The most accurate method to determine the transfer function \( H(f) \) is by structural testing\textsuperscript{R133-R134}. The best experimental estimate of \( H(f) \) is obtained by imposing a sinusoidal load \( M_0(t) \) at frequencies \( f_e \) to the system with a vibration exciter. For one excitation frequency two experiments are carried out: one where the excitation acts in the \( S_1 \)-plane and another in the \( S_2 \)-plane. The transfer function at frequency \( f_e \), \( H(f_e) \), is then obtained by solving the system of four equations:

\[
\begin{bmatrix} \hat{\tilde{M}}_0(f_e) \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} H_{11}(f_e) & H_{12}(f_e) \\ H_{21}(f_e) & H_{22}(f_e) \end{bmatrix}^{-1} \begin{bmatrix} \hat{\tilde{S}}_1(f_e) \\ \hat{\tilde{S}}_2(f_e) \end{bmatrix}
\]

Where “\( ^f \)” denotes Fourier transform and “\( ^\prime \)” indicates that the measured signals are different when applying the excitation load in the different planes.
This process is then repeated for different frequencies $f_e$ within the frequency band of interest and the frequency resolution is adapted during the process so the response at normal modes are resolved. Finally a curve-fitting technique can be used to improve the modal parameter estimation.

If the experimental set-up is carefully designed it can be approximated fairly well as a dynamic system with one degree of freedom (1-DOF), which facilitate the determination and description of the transfer function $H(f)$. This approximation is only valid if the system is isotropic, i.e. $H_{11}=H_{22}=H$ and $H_{12}=H_{21}=0$, and the eigenfrequencies of any higher order modes are far from the fundamental eigenfrequency $f_0$. If this is the case one can reconstruct the aerodynamic load components from the strain-gauge signals as:

$$\hat{M}_{a1,2}(f) = H^{-1}(f) \cdot \hat{S}_{1,2}(f)$$

An expression for the transfer function can be obtained from the characteristic differential equation for a system with 1-DOF, which is:

$$\ddot{S}(t) + 2\zeta \omega_0 \dot{S}(t) + \omega_0^2 S(t) = \frac{1}{m} M_0(t)$$

Where

$$\omega_0 = 2\pi f_0 = \sqrt{\frac{k}{m}}$$

is the fundamental eigenfrequency

$\zeta$ is the damping coefficient

$k$ is the system stiffness

$m$ is the system mass

Taking the Fourier transform of this equation yields the transfer function as:

$$H(f) \cdot k = \hat{S} \cdot \frac{k}{M_0} = \left[ 1 - \left( \frac{f}{f_0} \right)^2 + i2\zeta \frac{f}{f_0} \right]^{-1}$$

Thus the parameters to be experimental determined have been reduced to $f_0$, $k$ and $\zeta$.

The stiffness $k$ is determined from static calibration, i.e. by applying different static loads and measure the corresponding strain, as illustrated in Figure 54.

![Figure 54. Illustration of static calibration of the spring stiffness (from VOLVO S1 testing).](image)

The damping and the fundamental frequency are found from a dynamic calibration, i.e. a hammer test or applying a step function to the system or anything similar. In Figure 55 the measured response is shown when applying a step function to the VOLVO S7 nozzle.
The theoretical response to a step function to a 1-DOF system is:

\[ S(t) = S_0 e^{(-at)} \sin(2\pi f_d t + \varphi) \quad \text{where} \quad \begin{cases} f_d = f_0 \sqrt{1 - \zeta^2} \\ a = 2\pi f_0 \zeta \end{cases} \]

Thus, after analysis of the rate of decay, \( a \), and the oscillating frequency, \( f_d \), of the response signal, see Figure 56-Figure 57, the damping coefficient and fundamental frequency can be determined as:

\[ \zeta = \left[ 1 + \left( \frac{2\pi f_d}{a} \right)^2 \right]^{-1/2} \quad \text{and} \quad f_0 = \frac{f_d}{\sqrt{1 - \zeta^2}}. \]

Figure 55. Oscillating decay test with the VOLVO S7 nozzle.

Figure 56. Analysis of the decay rate of a response signal, example taken from the S7 test campaign.
Figure 57. Analysis of the oscillating frequency of a response signal using FFT example taken from the S7 test campaign.
8 SIDE-LOADS – PHYSICAL ORIGINS AND MODELS FOR PREDICTION

Side-loads have been observed both in sub-scale and full-scale rocket nozzles during transient operations like start-up or shut-down, as well as during stationary operation with separated flow inside the nozzle. Such forces, acting lateral to the main thrust direction are an undesired phenomenon, and may become a severe design constraint for new rocket engine concepts. The first important report dealing with side forces was published in the 1970’s within the frame of the J-2S testing.\textsuperscript{R95}

Potential origins for side-loads generated by an asymmetric wall pressure evolution inside the nozzle are:
- transition in separation pattern, FSS to RSS and vice versa.
- tilted separation line,
- pressure pulsations in the separation region and in the re-circulation flow region,
- aeroelastic coupling.

Furthermore, pressure pulsations acting from the outside on the nozzle shell, asymmetry arising from the manufacturing process, or asymmetric injection of propellant or/and film cooling gases may generate lateral forces, which however are not further considered in this discussion.

In the following we will discuss different models, developed on the basis of different origins.

8.1 SIDE-LOADS DUE TO TRANSITION IN SEPARATION PATTERN

It has been shown in sub-scale cold-gas experiments, that a transition in separation pattern from the free-shock separation (FSS) to the restricted shock separation (RSS) and vice-versa may occur.\textsuperscript{R90,R95} That these transitions also are the origin of two distinct side-load peaks was first shown by Östlund\textsuperscript{R85} based on his analysis of the VOLVO S1 test.

8.1.1 Origin of side load: observations of the VOLVO S1 nozzle flow

Figure 58 shows some typical measured steady-state wall pressure profiles in the VOLVO S1 nozzle during start up, i.e. the pressure ratio between the feeding and ambient pressure \( p_0/p_a \) is increased from 1 and upwards. As can be seen, the wall pressure profiles indicate FSS for \( p_0/p_a<15 \) and RSS for \( p_0/p_a>15 \) (cf. Figure 37). Hence, there is a transition of the flow separation pattern at \( p_0/p_a \approx 15 \). This transition from FSS to RSS can also be seen in the schlieren photos in Figure 83. To support the understanding of these schlieren photos, the corresponding calculated Mach number contours are shown in Figure 61. At \( p_0/p_a<15 \), the exhaust jet is seen to occupy only a fraction of the nozzle exit whereas at \( p_0/p_a>15 \) the exhaust is attached to the nozzle wall.

The measured wall pressure distributions during shut down are shown in Figure 59, where it can be seen that the transition between RSS and FSS occurs at a pressure ratio of about \( p_0/p_a=12 \), it is hence clear that there is a hysteresis effect. Figure 60 compares the wall pressure profiles at FSS and RSS condition at a pressure ratio of \( p_0/p_a=13 \). As can be seen the wall pressure distribution is quite different for the two cases. One of the most remarkable differences is that the RSS separation line is located much further downstream of the FSS separation line (the reason for this difference is explained in section 6.2). Thus, at the FSS to RSS and RSS to FSS transition inside the nozzle there is a considerable jump of the separation line, in connection with the RSS-FSS or RSS-FSS transition.
Figure 58. Wall pressure profiles in the VOLVO S1 nozzle during start up, see also Östlund et al\cite{8}.

Figure 59. Wall pressure profiles in the VOLVO S1 nozzle during shut down.

Figure 60. Comparison between wall pressure profile at FSS and RSS condition at $p_0/p_a=13$. 

63
Analysis of high-speed video recording of the schlieren pattern, see Östlund R 5, R 138, indicates that while the FSS flow field is moving towards the flow field transition point at $p_0/p_a=15$, the free jet experiences a few cycles with a tilted separation plane. Next an unsteady asymmetrical reattachment of the flow occurs as the feeding pressure is increased further, successively occupying more and more of the nozzle wall, until the entire flow is reattached, i.e. the RSS state prevails. In this state, the degree of asymmetric separation is small and the flow is quite stable and only small fluctuations of the flow can be observed as a compression wave (or possible a secondary separation bubble) is ejected from the nozzle exit lip at $p_0/p_a=23$. However, when the reattachment point reaches a position close the nozzle exit, the flow begins to pulsate with a frequency of about 100 Hz. This happens at a pressure ratio of 25. This is caused by a sudden increase of the plateau pressure behind the separation shock, which occurs when the enclosed recirculating zone is opened up and ambient air is sucked in to the nozzle. The increase in pressure, forces the separation point to move upstream again, once more closing the recirculating zone. The cycle repeats itself until the feeding pressure is sufficiently large to move the reattachment zone completely out of the nozzle, which in this case occurs at a pressure ratio of 30. When decreasing the feeding pressure, this procedure is repeated, however, now in the reversed order. Large fluctuations of the flow field can be observed as the pressure ratio is reduced below 28. At this pressure ratio, the separation line has moved into the nozzle and a repeated closing and opening of the recirculating zone takes place, until a stable reattachment is achieved at $p_0/p_a=26$. Between $p_0/p_a=26-13$, the flow field is again stable showing only small fluctuations at $p_0/p_a=23$, 21 and 20, which corresponds to instants when compression waves is moving into the nozzle. At $p_0/p_a=12$ the flow becomes unsteady and the stripy pattern, see Figure 83, characterising RSS suddenly disappears.

Figure 62 shows a time record of the measured side load torque during a start up and shut down process. Two distinct load peaks can be identified both during start up and shut down. The main characteristic of these side forces is their high value and the impulsive occurrence. In Figure 63 and Figure 64 these data are given in terms of percent of peak measured loads versus the feeding to ambient pressure ratio for the start up and shut down transient respectively.

---

1 For the computations the standard Wilcox $k$-$\omega$ model with an ad hoc realizability correction, $\mu=\min(\mu,0.1)$, was used together with the same computational model and type of grid as presented in section 11.3. More details of the computations can be found in R 138- R 139.
As indicated in the figures it is one significant load peak at a pressure ratio \( p_0/p_a \approx 15 \) and second at a pressure ratio of 28 during the start transient. Corresponding side load peaks during the throttle down occurs at pressure ratio of 12 and 28. It is thus obvious that these side load peaks is coupled to the transitions from FSS to RSS or vice versa as described above.

Figure 62. Side-loads due to transition in separation pattern, also published in R 5

Figure 63. Normalised side load torque vs. feeding to ambient pressure ratio during start up, also published in R 5.

65
Figure 64. Normalised side load torque vs. feeding to ambient pressure ratio during shut down, also published in R 5.

The above observations and conclusion, which has been partly published in reference R 5, was the ignition for intensive research both within and outside Europe. Further sub scale experiments performed within different FSCD test campaigns and recent Japanese experiments confirmed this mechanism for TOP and CTIC nozzles both featuring an internal shock. In addition, test results of the Vulcain 1 engine confirmed this mechanism as key driver during both start-up and shut-down.

8.1.2 Side-load model

By assuming that the initial transition from FSS to RSS requires a certain time, a phase might exist during which one side of the nozzle experiences a free shock separation while at the other side the flow reattaches. Since the separation point is located further downstream in the restricted shock case and the wall pressure behaviours are totally different between the two cases, severe lateral forces are acting on the nozzle. The main characteristic of these side forces is their high value and the impulsive occurrence as shown above.

Based on the briefly described restricted shock separation models of reference R 3 and R 6, see chapter 6.2, Dasa/DLR and Volvo have developed side-load models by assuming that the initial transition from free-to restricted shock separation is the key side-load driver in TOP and CTIC rocket nozzles. The basic idea behind is, that at the instant of transition a maximum side-load is expected if one half of the nozzle features FSS-, while the other half shows already RSS flow condition. For this case, the side-load calculation is squarely based on physical reasoning namely from a momentum balance across the complete nozzle surface, as illustrated in Figure 65.

With this model the aerodynamic side-load can be calculated. Due to the short duration of the aerodynamic side-load, pulse excitation theory can be used when evaluating the mechanical load.

For a single pulse excitation, the dynamic response factor (i.e. the amplification of the applied load due to the dynamic system) is always less than 2. The most critical pulse is the single square wave, since it contains the highest energy that any single pulse can have. Figure 66 shows the Shock Response Spectrum (SRS) for a single square wave. A further example of waves is the half-sine wave with its SRS depicted in Figure 67. The half-sine and the triangular pulse, see Figure 68, are often good approximations to actual pulse shapes, e.g. the pulse creating the side-load when the separation pattern is changed from FSS to RSS. With the
knowledge of the transition time, $t_1$, and the mechanical eigenfrequency, $\omega = 2\pi/\tau$, the dynamic response factor can be obtained from Figure 66-Figure 68.

In Table 4, values with this side-load model are compared with the maximum measured values in different subscale and full-scale experiments showing that the accuracy is within 6%.

<table>
<thead>
<tr>
<th>Nozzle</th>
<th>$p_{0,m}/p_{0,c}$</th>
<th>$M_m/M_c$</th>
</tr>
</thead>
<tbody>
<tr>
<td>VolvoS1</td>
<td>0.94</td>
<td>1.01</td>
</tr>
<tr>
<td>VolvoS3</td>
<td>1.0</td>
<td>1.02</td>
</tr>
<tr>
<td>Vulcain</td>
<td>1.05</td>
<td>1.05</td>
</tr>
</tbody>
</table>

Table 4. Comparison between VAC calculated (subscript c) and measured (subscript m) transition feeding pressure ($p_0$) and aerodynamic torque (M), also published in Östlund et al. 108.

Figure 65. Asymmetric flowfield inside nozzle at instant of FSS-RSS transition for worst case side-load prediction. Control surface for momentum balance included. Momentum of impinging jet on wall taken into account at $x_w$, also published in R 105.

Figure 66. Shock response spectrum (SRS) for a single square wave, $t_1=\text{pulse duration time}$, $\tau=\text{period time}$, also published in Östlund et al. 108.
8.2 SIDE-LOADS DUE TO TILTED SEPARATION LINE

The assumption of a tilted separation line under pure free shock conditions is the basis of several side-load models, e.g. of Pratt and Whitney, Rocketdyne, Aerojet and Schmucker\textsuperscript{94}. The principle of this basic idea is illustrated in Figure 69.
The side load force acting on a nozzle wall is determined by:

\[ F_{\text{sl}} = \int_{x_{\text{sep}}}^{x_{\text{sep}w}} \int_{0}^{2\pi} (p_a - p_w) \cos \tau \, d\tilde{A} \]

Eq. 38

Where \( d\tilde{A} \) is a nozzle surface element and \( \tau \) is the local contour angle.

If the wall pressure distribution is axisymmetric, no side loads are present, however, if an asymmetry appears, as in Figure 69, a side force arises given by integration over the separation region, i.e.:

\[ F_{\text{sl}} = \int_{x_{\text{sep}}}^{x_{\text{sep}w}} \int_{0}^{2\pi} (p_a - p_w) \cos \tau \, d\tilde{A} \]

Eq. 39

where \( x_{\text{sep}} \) and \( x_{\text{sep}w} \) are the distances at which the asymmetric flow separation occurs. This equation can be written in a simplified form as:

\[ F_{\text{sl}} \approx (p_a - p_{\text{sep}}) A_{\text{sl}} \]

Eq. 40

Where \( A_{\text{sl}} \) is the projection of the region with asymmetric flow separation on a plane perpendicular to the side force direction.

Knowing the distance between the point where the side force is applied, \( l_{\text{sl}} \), and the nozzle gimballing point, the force acting on the engine actuator denoted by subscript ac can be determined as:

\[ F_{\text{ac}} = F_{\text{sl}} \frac{l_{\text{sl}}}{l_{\text{ac}}} \]

Eq. 41

In order to predict the magnitude of the side loads, the wall pressure difference and the area of the region with an asymmetric flow separation needs to be known. The wall pressure difference can be predicted with a flow separation criterion, whereas in order to determine a zone of asymmetric flow separation, it is necessary to have a model of this phenomenon.
Schumcker made the following conclusions concerning the character of the side loads based on the experimental work performed by Nave and Coffey with the J2-S engine and its sub-scale models.

- The value and direction of the side loads are unsteady.
- The value of the side loads depends on the area of the asymmetric flow separation zone.
- The side loads are amplified due to dynamic effects.
- The stability of the separated flow increases with increasing nozzle contour angle.
- The value of the side loads decreases with increasing wall pressure gradient.
- Zero or negative wall pressure gradient leads to very high side loads.
- The maximum side loads are obtained at chamber pressures 10-20% below the minimum chamber pressure providing attached flow.

He further assumed that:

- The main part of the side load is generated due to a bias of the separation line from its averaged symmetric position; therefore the separation point is the location where the side force is applied.
- Fluctuations of the separation zone are caused by pressure fluctuations arising due to:
  - pressure oscillations in the combustion chamber
  - recirculation of the atmospheric air
  - chemical reactions in the boundary layer

Schmucker then proceeded to construct a method, where the aerodynamic side-load, $F_{sl}$, due to the unsymmetrical separated area, $A_{sl}$, is calculated as:

$$F_{sl} = (p_u - p_t)A_{sl} = (p_u - p_t) \cdot b(x_{sepo} - x_{sepu}) = (p_u - p_t) \cdot b\Delta l_{fl} \quad \text{Eq. 42}$$

where $\Delta l_{fl}$ is the length of the unsymmetrical separated area and $b$ is a measure of the effective asymmetry of the separation zone in the circumferential direction. In order to calculate $\Delta l_{fl}$ Schmucker made the assumption that the fluctuations, and hence the unsymmetrical pressure release at the wall are proportional to the nominal wall pressure $p_w$, i.e.:

$$\frac{\Delta p_w}{p_c} = K_{fl} \frac{p_w}{p_c} \quad \text{Eq. 43}$$

By applying a separation criterion, $\Delta l_{fl}$ can then be obtained from the angle of intersection between the curves enclosing the wall pressure fluctuation and the separation criterion, as shown in Figure 70.
\[
\Delta l_{fl} = \frac{\Delta p_w}{dp_w} - \frac{dp_w}{dx} = K_{fl}r_i \frac{p_w}{p_e} \frac{1}{d(x/r_i)} \frac{d(p_w/p_e)}{d(x/r_i)} \frac{p_a}{d(x/r_i)} \frac{d(p_a/p_e)}{d(x/r_i)} \text{ Eq. 44}
\]

After some manipulation and the use of the Schmucker separation criterion, Eq. 30, one can write:

\[
\frac{d(p_w/p_e)}{d(x/r_i)} = \frac{d(p_w/p_e)}{d(x/r_i)} \frac{1+(\gamma-1)/2M_i^2}{1+(\gamma-1)/2M_i^2} \frac{1.2032}{1.2032} \text{ Eq. 45}
\]

Thus, one obtains:

\[
\Delta l_{fl} = K_{fl}r_i \frac{p_w}{1-(1+(\gamma-1)/2M_i^2)} \frac{1}{d(x/r_i)} \frac{1}{d(x/r_i)} \frac{1}{d(x/r_i)} \frac{1}{d(x/r_i)} \text{ Eq. 46}
\]

The value of the fluctuation coefficient needs to be determined from test data, e.g. Schmucker found an average value of \(K_{fl} = 0.05\) for the J2-D engine.

The shape of the unsymmetrical separation line is described by the coefficient \(K_{s}\) as:

\(b = 2rK_s\)

For this shape coefficient, which cannot exceed 1, the following values apply:

\(K_s = 1\) : unsymmetry at 180° of nozzle circumference (maximum side-load).
\(K_s = \pi/4\) : inclined separation line.
\(K_s = 0.3-0.4\) : effective value, used as default values in the Schumcker model.
\(K_s = 0\) : symmetrical separation line.

With the above equations, the aerodynamic side-load can finally be expressed as:

\[
F_{sl} = 2K_s K_{fl}r_i r_i^2 \frac{p_i}{p_a} \frac{p_a}{p_e} \frac{1}{d(p_w/p_e)} \frac{1}{d(x/r_i)} \frac{1}{d(x/r_i)} \frac{1}{d(x/r_i)} \text{ Eq. 47}
\]

As the value \(\Delta l_{fl}\) is proportional to \((dp/dx)^{-1}\), the wall pressure gradient is the most important factor that influences the value of the side load. When the pressure gradient increases, the side load decreases. When the separation location moves towards the nozzle exit, the pressure gradient decreases. This will thus result in an increase of the side load as \(\Delta l_{fl}\) increases. When the forward boundary of the separation fluctuation zone has reached the nozzle exit, the side load has its maximum value. In the further motion of the separation zone the fluctuation zone decreases too finally disappear. By that Schmucker explains the qualitative dependence of the side force as function of the chamber pressure.

A comparison of the model results with the side forces measured in the J-2S engine is shown in Figure 71.
Schmucker’s model is based on an analysis of experimental data obtained from tests with J-2S engine and J-2S sub-scale model tests. The logic of this model construction is faultless. Nevertheless, a disagreement between “hot” and “cold” experimental data indicates that the physical treatments of some elements of the model are incorrect. A model intended for the determination of the fluctuation of the separation zone is the main element of the Schmucker model. The value of $\Delta l_{fl}$ predicted with the model is significantly greater than the boundary layer thickness and does not connect with it in any way. Besides, the zero pressure gradient is a singular point with Schmucker’s method.

Experiments, see section 5.2.3 and 6.3.1, have shown that when the flow separates intense pressure pulsations are observed in the separation zone. The length of this zone is connected with the boundary layer properties just before the separation point and is e.g. equal to several (2-3) boundary layers thickness’ in obstacle induced separation. Experiments of nozzle flow separation conducted by VOLVO validate these conclusions of the length of the separation zone and the wall pressure pulsations, see Figure 42 and R 108.

However, there is a possibility for another physical phenomenon, which can not manifest itself in “cold” experiments. That is a possibility for the existence of chemical reactions in the separation zone between the hydrogen incompletely burned in the near-wall region and the atmospheric air entering from the recirculating zone. This phenomenon has a chaotic nature and causes additional pressure disturbances which increases the length of the fluctuating separation zone.

The side load models of Pratt and Whitney, Rocketdyne, Aerojet is similar in the form to the Schmucker model. However, these models essentially differ from the Schmucker model in both physical and logical bases. According to the current author’s opinion, these methods have no physical logic. Besides, they are based on dispersion of incorrectly generalised experimental data of flow separation.
8.3 SIDE-LOADS DUE TO RANDOM PRESSURE PULSATION

Random oscillation of the separation line and random pressure pulsation in the separated flow region are the basic idea of the Dumnov side-load model. The method is based on a statistical generalisation of empirical data for the pulsating pressure field at the wall. The empirical data are mainly based on sub-scale cold-gas experiments with separated nozzle flows. For these experiments, only conical and truncated ideal nozzles were used.

The Dumnov model can be outlined as follows.

The momentary side force acting on the nozzle wall from the gas is:

$$F'_s = \int p'_w(x) \cos \varphi \, dx \, d\varphi$$  \hspace{1cm} \text{Eq. 48}

with the corresponding autocorrelation as:

$$K_F(\tau) = 2\pi \int_0^\pi \int_0^\pi K_p(\tau, x, \xi, \Delta \varphi) r(x) r(\xi) \cos \Delta \varphi \, d\xi \, dx \, d\varphi$$  \hspace{1cm} \text{Eq. 49}

where

$$K_F(\tau) = \langle F'_s(t) \cdot F'_s(t + \tau) \rangle$$

$$K_p(\tau) = \langle p'_w(x, \varphi, t) \cdot p'_w(\xi, \varphi - \Delta \varphi, t + \tau) \rangle$$

and $\langle \ldots \rangle$ denotes the statistical averaging.

$K_p$ gives the intercorrelation function of the pressure pulsation at the nozzle wall. The correlation function can be transformed into the spectral density of the side force using Fourier transform, i.e.:

$$W_f(f) = 2\pi \int_0^\pi \int_0^\pi W_p(f, x, \xi, \Delta \varphi) r(x) r(\xi) \cos \Delta \varphi \, d\xi \, dx \, d\varphi$$  \hspace{1cm} \text{Eq. 50}

From the above equations we can see that the key to determine the side-load is the interspectral density of the pressure fluctuations, $W_p$, acting on the nozzle wall. The determination of $W_p$ is a fairly difficult problem and Dumnov solved this by generalising test data in the following form:

$$\overline{W}_p = W_p \cdot \frac{U}{\sigma_p \theta_i}$$  \hspace{1cm} \text{Eq. 51}

Where $U$ is the velocity of the separated jet, $\sigma_p$ is the rms level of the pressure pulsations, $\theta_i$ is the momentum thickness just before the start of the interaction.

Dumnov neglected the pressure fluctuations in the boundary layer of the attached flow, since they are substantially smaller than the ones in the separation point region, $x_t \leq x \leq x_p$, and the recirculating flow region, $x_p < x < L$, cf. Figure 31. Thus, the side force in the Dumnov model is only governed by the pressure fluctuations in the latter two regions.

According to Dumnov, the rms value of the pressure fluctuation in the separation point region can be evaluated by assuming a sinusoidal fluctuation between the two pressure levels $p_i$ and $p_p$, i.e.:

$$\sigma_{sh} \approx \frac{p_p - p_i}{2\sqrt{2}}$$  \hspace{1cm} \text{Eq. 52}

By introducing the Mach number at $x=x_t$ and assuming an isentropic expansion...
\[
\pi(M_i) = \frac{p_i}{p_0} = \left(1 + \frac{\gamma - 1}{2} M_i^2 \right)^{\frac{\gamma}{\gamma-1}}
\]
Eq. 53

The rms value can be calculated as:

\[
\sigma_{sh} = \frac{p_0}{2^{\frac{\gamma}{\gamma-1}}} \left(1 + \frac{\gamma - 1}{2} M_i^2 \right)^{\frac{\gamma}{\gamma-1}} \frac{p_p/p_i - 1}{2\sqrt{2}}
\]
Eq. 54

For calculation of the rms pressure level in the recirculating zone, Dumnov proposes the following expression:

\[
\sigma_{rec} = \frac{p_p}{2^{\frac{\gamma}{\gamma-1}}} \left(1 + \frac{\gamma - 1}{2} M_i^2 \right)^{\frac{\gamma}{\gamma-1}} \frac{d\pi(M_i)}{dM_i} + 1 \frac{d}{dM_i} \frac{d\pi(M_i)}{dM_i} dM_i \frac{L_{sh}}{\pi_{sh}}
\]
Eq. 55

Which is derived by differentiating the relation \(p_{crit}=p_p/p_i\). Here \(L_{sh}\) is the length of the interaction region i.e., \(L_{sh}=x_p-x_i\).

The local and mutual spectral densities were then obtained at various operating modes of a selected TIC nozzle and normalised according to eq. 51. From there, the side force in different nozzles induced by the gas can then be determined.

The obtained aerodynamic side force is translated into a mechanical load on the test stand or rocket by use of a transfer function, \(H(f)\), which characterise the mechanical system.

The spectral density of the mechanical system oscillations, \(W_M\), can then be expressed as:

\[
W_M(f) = |H(f)|^2 W_F(f)
\]
Eq. 56

The total rms level of the mechanical side force then becomes:

\[
\sigma_Q \approx F_{sl} = \sqrt{\int_0^\infty W_M(f) df}
\]
Eq. 57

The application of the Dumnov-model to the Russian rocket nozzle RD-0120 gives reasonable agreement between measured and predicted side-load\(^R^{101}\), as can bee seen in Figure 72. In reference R 101, the mechanical system representing the test stand with the RD-0120 nozzle is basically a 1-degree of freedom harmonic oscillator.
The basic idea of the Dumnov side-load model, i.e. a random oscillation of the separation line and random pressure pulsation in the separated flow region are correct and in agreement with experimental observations found in experiments of obstacle induced separation, see section 5.2.4. The interaction length is one of the main elements in this type of side load model and a correct value of $L_{sh}$ is thus essential. Dumnov gives no information of the interaction length in his paper. However, he probably used experimentally determined values of the interaction length when calibrating the model. When applying it to other nozzles, Dumnov uses a semi-empirical correlation function for the interaction length similar to that derived from the free interaction theory, see Figure 42, where the interaction length is coupled to the incoming boundary layer properties. Since no detailed information has been given of this interaction length correlation, except that $l/\theta = f(M_s, T_w)$ according to Dumnov et al., the validity and the influence of this correlation can not be assessed. Nevertheless, the approach is superior to the Schmucker model, where the interaction length has no coupling to the boundary layer properties at all.

The dumnov approach was tested by applying the model on data obtained in tests with the LEA TIC nozzle by Girard and Alziary. This yields a constant rms level of the pressure fluctuation in the separation point region of $\sigma_{\theta}/\sigma_{\theta}' = 83$, in fair agreement with max values of the test data, see Figure 73. The Dumnov approach predicts further one constant fluctuation level $\sigma_{\text{rec}}/\sigma_{\text{rec}}' = 17$ in the recirculating region, whereas the measurements indicates a decreasing rms level, from $\sigma_{\text{rec}}/\sigma_{\text{rec}}' = 20$ to $\sigma_{\text{rec}}/\sigma_{\text{rec}}' = 12$, with increasing distance from the interaction region. A more accurate method to determine the pressure fluctuations in the interaction domain on physical basis would be to use the Kistler approach, see section 5.2.4. In contrast to the Dumnov model, which only gives one constant value of $\sigma_{\theta}'$ throughout the interaction region, the Kistler approach renders the streamwise evolution of $\sigma_{\theta}'$. According to Kistler the mean-square pressure fluctuation around the mean pressure in the interaction zone can be expressed as (cf. Eq.27):

\[ \sigma_{\theta}' = \sigma_{\text{rms}} \]

The difference in the measured values in the interaction region between the two tests is an effect due to the operational conditions, which were not exactly identical in the two tests. This influences the start and the extension of the interaction region, see Figure 73.
Where \( \sigma p'_{i}^{2} \) and \( \sigma p'_{p}^{2} \) is the mean-square pressure fluctuation in the unperturbed boundary layer upstream the interaction and in the plateau pressure region respectively. \( \varepsilon \) is the fraction of time that the plateau pressure region is acting over the point of interest, i.e. an “intermittence” factor. According to Erengil and Dolling\(^{61}\) the error function shows a good fit to the intermittence factor \( \varepsilon \), which means that the location of the separation shock has a Gaussian distribution within the intermittent region.

The rms distribution and the corresponding level of \( \sigma p' \) spatial average over the interaction domain, obtained with the Kistler method is included in Figure 73. As can be seen the calculated rms curve fits the measured values surprisingly well. Figure 74 shows the change of the rms level in the short VOLVO S7 nozzle as the interaction zone moves upstream during down ramping of the chamber pressure. The figure shows this behaviour for two different pressure transducers located at axial positions corresponding to \( M_{\infty}=3.8 \) and \( M_{\infty}=4.1 \) respectively. As can be seen the curves are very similar for the two positions. The interaction zone moves over the transducers during the time \( t_{p}-t_{i} \), where subscript \( i \) and \( p \) refers to the start of the interaction and the plateau point respectively. Since the global movement of the separation front is slow and almost constant and the changes in \( p_{i} \) and \( p_{p} \) are modest during the down ramping, the obtained curves can be interpreted as the streamwise evolution of \( \sigma p' \), i.e. \( (t-t)/\langle t_{p}-t_{i} \rangle=s \). When comparing Figure 73 with Figure 74 it can seen that the streamwise distribution of the rms levels obtained with the Kistler approach also corresponds well with test data obtained during this transient operation of the VOLVO S7 nozzle.

Based on the experimental evidence we can see that \( \sigma p'_{i} \) and \( \sigma p'_{p} \) are small compared to \( (p_{p}-p_{i}) \), thus a reasonable approximation of the Kistler expression is:

\[
\sigma p'_{i}^{2} \approx \varepsilon \left[ 1-\varepsilon \right] \left( p_{p}-p_{i} \right)^{2}
\]

According to this expression, the maximum rms, \( \sigma p'_{\text{max}} \), occurs at \( \varepsilon=0.5 \) (i.e., the mid-point of the intermittent region) and has a value of \( \sigma p'_{\text{max}} = 0.5 \left( p_{p}-p_{i} \right) \). Equation 59 then gives the corresponding average rms value in the intermittent region \( \langle \sigma p' \rangle = 0.2420 \left( p_{p}-p_{i} \right) \). This is close to the averaged value \( \sigma p' = 0.2514 \left( p_{p}-p_{i} \right) \) obtained by inserting \( \sigma p'_{i} \) and \( \sigma p'_{p} \) from the LEA TIC test. The averaged rms level obtained with the Dumnov approach is significantly higher compared with the one obtained with the Kistler approach. The difference of the averaged rms level between the two approaches is \( \sigma p'_{\text{Dumnov}}/\sigma p'_{\text{Kistler}} \approx \sqrt{2} \). This implies that the assumption of a sinusoidal fluctuation between the two-pressure levels \( p_{i} \) and \( p_{p} \) done by Dumnov is too simple and over predicts the averaged fluctuation level in the intermittent region.
Figure 73. Rms pressure fluctuations in the LEA TIC nozzle, comparison between measured and calculated values, (Test data taken from Girard and Alziary®).

Figure 74. Integrated rms values of pressure signal at two different axial locations in the VOLVO S7 short nozzle during down ramping of $p_c$. Each value based on 800 samples = 0.2 [s].
Analysis of the expression of the rms pressure level in the recirculating zone proposed by Dumnov, equation 55, leads to:

$$\sigma_{rec} = \frac{p'_p}{\sqrt{2}} = dp_p \frac{L_{sh}}{dx} \frac{\sqrt{2}}{2\sigma'}$$

Eq. 60

Which means a sinusoidal fluctuation with amplitude $p'_p$. Hence it is assumed that purely the pressure behind and the motion of the separation shock causes the pressure fluctuation level in the recirculating zone. As the mean pressure in the interaction region is a result of the intermittence of the flow, i.e. the separation shock moving back and forth see Figure 32, the gradient of the mean pressure is a kind of measure of this motion. Thus the proposed expression as a first order of approximation of the fluctuation at the plateau point seems to be a good assumption especially when comparing it with test data. The decrease of the rms level in the recirculating region as indicated in the measurements might be interpreted as a damping of the fluctuation level originating from the shock motion as the distance from the interaction region is increased. The calculated value with the Dumnov approach is thus conservative when used as a mean rms value over the entire recirculation region.

In general it can be said that appropriate parameters for normalising intermittent region power spectra are an area requiring substantially more work. However, Gonzalez and Dolling\textsuperscript{118} showed that the spectra in the intermittent region, generated by different diameter cylinders, collapsed when plotting the magnitude $W_p(f) \cdot f / \sigma_p^2$ versus a reduced frequency given as $f L_{sh} / U_\infty$. This scaling using the length of the of the intermittent region, $L_{sh}$, appears to have a firm foundation, but the use of the free stream velocity just before the onset of the interaction, $U_\infty$ should be viewed as tentative, since all of the experiments were carried out at the same $U_\infty$. Dumnov uses $U / \sigma_p^2 \theta$ for normalising the magnitude of the experimental determined spectral and interspectral characteristics of the pressure pulsations whereas no information is given of how and if the frequency axis is normalised. This together with the lack of experimental data in the paper by Dumnov makes it impossible to estimate how generalised the spectrum obtained actually is.

One of the main advantages with the Dumnov approach is that the structure dynamic characteristic of the rocket engine or the model facility is taken into account in a more psychical manner compared to the common approach with a constant dynamic response factor used in older models. Even if the amplification in the Dumnov model is only accounted for through a transfer function characterising a 1DOF system, it was a significant improvement in the methods of side-load modelling at the time it was published. It gave the engineer a valuable tool to estimate expected side-loads, needed to mechanically define the thrust chamber structure to ensure mechanical integrity under worst case condition and to design gimbaling system dampers etc such that the amplified load is kept within reasonable limits.

Finally, it must be stressed again that the overall logic of the Dumnov approach is correct. However, as indicated in the above analyses corrections and improvements could be done. The development and validation work of such improved models is currently ongoing at the different partners of the FSCD group. E.g. at Volvo Aero\textsuperscript{108}, a model is under development to calculate this type side-loads due to random pressure fluctuations. Figure 75 shows a comparison between a first prediction with this model and experimental side-load data for the S7 nozzle. As can be seen, the general trend of the side-load is captured well, whereas there are some deviations in the prediction of its magnitude.
8.4 SIDE-LOADS DUE TO AEROELASTIC COUPLING

A possible reason for side-loads is the aeroelastic interaction between flow induced wall-pressure fluctuations and the mechanical eigenmodes of the nozzle and thrust chamber. Slight variations in wall pressure may cause significant distortion of the contour, which in turn results in a further variation in wall pressure and the system forms a closed loop, which may result in a significant amplification of the initial load. The study of these closed-loop effects in separated nozzle flows is rather complex, requiring dynamic models of the mechanical nozzle-engine support system, the flow separation, as well as the coupling between these two. However, a simple technique for handling these difficult coupling problems has been proposed by Pekkari\textsuperscript{102,103}. The model consists of two main parts, the first dealing with the equation of motions of the thrust chamber as aerodynamic loads are applied, and a second part modelling the change of the aerodynamic loads due to the distortion of the wall contour. The wall pressure in the attached region is the nominal vacuum pressure profile with a pressure shift due to the displacement of the wall. In the original work by Pekkari, this pressure shift is determined with the use of linearised supersonic flow theory. However, experience has shown that this theory tends to overpredict the pressure shift and Östlund\textsuperscript{108} therefore proposed a modified approach where the pressure shift is extracted from 3D Euler simulations. The separation line in the nozzle is assessed with a simple separation criterion of Summerfield type (cf. Eq.28). The wall pressure in the separated region is assumed to be equal to the ambient pressure. This model predicts the aeroelastic stability, the modification of eigenfrequencies due to aeroelastic effects, as well as the transient behaviour during start up and shutdown of the nozzle. Different mechanical eigenmodes can be treated, however, from side-load point of view, the aeroelastic behaviour of the bending mode is the most relevant one. In the following paragraphs, the model is outlined for the first bending mode, simplified as a pure bending around a flexible joint or cardan located at the throat. The model is also compared with VOLVO S1 and S6 sub scale test data.

8.4.1 Aeroelastic analysis

In the following section we will describe the aeroelastic theory applied to VOLVO S1 and S6 test cases. We consider the geometry for the flow and the nozzle wall motion as indicated in Figure 76.
In order to simulate the first bending mode the nozzle is mounted on a flexible joint or cardan with stiffness $k$ located at the throat. $\theta$ is the tilt angle between the nozzle centre line and the combustion chamber centre line. $L$ is the length (from the throat to the exit), $m$ is the mass, $J_y$ is the mass of inertia around the y-axis, $\tau$ is the local contour angle, and $r$ is the local radius of the nozzle. $w$ is the displacement of the nozzle wall. The azimuthal location is denoted by $\phi$.

Following the analysis of Pekkari,¹⁰² R¹⁰³, the system is considered as quasi static with respect to the flow, i.e. the characteristic time scales of the flow are considered to much larger than the characteristic time scales of the mechanical system. The equation of motion in the $y$-direction for the bending of the nozzle by an angle $\theta$ is:

$$J_y \ddot{\theta} = M_m(\theta) + M_a(\theta, P_{cc})$$  \hspace{1cm} \text{Eq. 61}$$

At equilibrium,

$$M_m(\theta) + M_a(\theta, P_{cc}) = 0$$  \hspace{1cm} \text{Eq. 62}$$

Here $M_m$ is the mechanical torque, i.e the restoring torque of the spring in the nozzle suspension, which for small displacements is:

$$M_m(\theta) = -k\theta$$  \hspace{1cm} \text{Eq. 63}$$

and $M_a$ is the aerodynamical torque induced by the pressure load onto the nozzle wall, i.e.:

$$M_a(\theta, P_{cc}) = \iint \hat{x} \times [p_a(\hat{w}(\theta), x, P_{cc}) - p_a] \cdot \hat{n} \, dS$$  \hspace{1cm} \text{Eq. 64}$$

where $\hat{n}$ is the wall surface normal vector facing the flow and $\hat{x}$ is the corresponding vector of location:

$$\hat{n} = \{ -\sin \tau, \cos \tau \cos \phi, \cos \tau \sin \phi \}$$  \hspace{1cm} \text{Eq. 65}$$

$$\hat{x} = \{ x, r(x) \cos \phi, r(x) \sin \phi \}$$  \hspace{1cm} \text{Eq. 66}$$

Upstream of the separation point, $x_{sep}$ (the point of minimum static wall pressure is usually indexed “$sep$” or “$i$” although the physical separation occurs later), the pressure is the free stream pressure, $p_a$. Downstream
of the separation point a pressure recovery occurs and the pressure gradually approaches the ambient pressure. For simplicity, the pressure downstream of the separation is here set equal to the ambient pressure, \( p_a \), i.e.

\[
p_w(\bar{w}, \bar{x}, P_{cc}) = \begin{cases} 
  p_{cc} \left( \frac{p_{w,0}(x)}{P_{cc}} + \Psi(\bar{w}, \bar{x}) \right) & ; \quad x \leq x_{sep} \\
  p_a & ; \quad x > x_{sep}
\end{cases}
\]  

Eq. 67

Here \( p_{w,0}(x) \) denotes the vacuum pressure profile in the undeflected nozzle. The second term in the pressure upstream of separation is the disturbance of the free stream pressure due to a small wall displacement, i.e.

\[
\Psi(\bar{w}, \bar{x}) = \frac{p_w(\bar{w}, \bar{x}, P_{cc}) - p_{w,0}(x)}{p_{cc}}
\]

Eq. 68

The location of the separation point is given by a separation criterion of Summerfield type (cf. Eq. 28):

\[
p(x_{sep}) = p_{sep} = 0.3p_a
\]

Eq. 69

In the original work by Pekkari, the pressure shift, \( \psi \), was calculated with the use of the small perturbation theory (SPT) (cf. Eq. 19-20), i.e.:

\[
\Psi(\bar{w}, \bar{x}) = \frac{\rho_{w,0}[\bar{M}_{w,0}^2 - 1]}{p_{cc}} \frac{\partial w}{\partial s} = B' \frac{\partial w}{\partial s}
\]

Eq. 70

Here \( w = \bar{w} \cdot \bar{n} \) denotes the normal displacement of the nozzle wall surface facing the flow and \( s \) is the arclength along the wall in the axial direction. \( B' \) is the normalised pressure shift coefficient, which expresses the change in pressure with the wall slope.

However, experience has shown that SPT tends to overpredict the pressure shift in cases where 3D effects are significant. A modified approach was therefore proposed by Östlund\textsuperscript{108} where the pressure difference \( B' \) is directly extracted from 3D Euler simulations:

\[
B'(x) = \frac{\Psi(\bar{w}, \bar{x})}{\partial w/\partial s} = \frac{p_w(\bar{w}, \bar{x}, P_{cc}) - p_{w,0}(x)}{p_{cc} \partial w/\partial s}
\]

Eq. 71

This can be seen in Figure 77, where the measured and the calculated wall pressure profile are shown for the undeflected and deflected (1°) S1 nozzle, respectively. The improvement obtained with CFD/Euler is obvious and is more pronounced in corresponding normalised pressure shift coefficient \( B' \), shown in Figure 78. As indicated in the figure, the SPT method overpredicts the pressure shift coefficient by approximately a factor 4 for this case. The CFD predictions on the other hand show close agreement with the experimental data and thus validate the use of Euler simulations for calculating the pressure shift coefficient.
Figure 77. Measured and calculated wall pressure in the S1 nozzle static deflected 1 degree, also published in Östlund et al.\textsuperscript{R108}.

It must be emphasised that the deviation of the wall pressure due to bending around the throat is highly dependent on the nozzle contour itself. As shown in reference R\textsuperscript{110} and R\textsuperscript{119}, the induced compression and expansion waves in a bent 15° conical nozzle interacts such that the pressure deviation trend is actually reversed, cf. Figure 79 and notice the negative value of $B'$. It was observed that on the side that was deflected...
away from the flow, where more expansion is expected, the wall pressure was in some portions of the nozzle even higher than on the opposite side, which was deflected into the flow. This finding has been confirmed by own numerical simulations, see Figure 79, and underlines the necessity of case-sensitive methods.

![Figure 79. Normalised pressure shift coefficient in conical nozzle, also published in Östlund et al.](image)

To proceed with the aeroelastic stability analysis, we next expand the free stream pressure around the initial location of the separation line, \( x_{sep0} \)

\[
p_{-0}(x_{sep}) = p_{-0}(x_{sep0}) + \frac{dp}{dx}(x_{sep} - x_{sep0}) + \ldots
\]

Eq. 72

Equation 67 written at the axial station \( x_{sep} \) is:

\[
p_{-}(\vec{w}, \vec{x}, P_{e}) = p_{-0}(x) + P_{e} B' \frac{\partial w}{\partial s} ; x = x_{sep}
\]

Eq. 73

The separation pressure \( p_{\infty} \) at \( x=x_{sep} \) approximated for the deformed wall contour by equation 73, will be the same as the separation pressure \( p_{\infty}(x_{sep0}) \) for the undeformed wall contour included in equation 72, which can be written as:

\[
p_{\infty}(x_{sep0}) = p_{\infty} at x=x_{sep}
\]

Eq. 74

and hence equation 73 and 72 give:

\[
x_{sep0} - x_{sep} = \frac{P_{e} B' \frac{\partial w}{\partial s} \bigg|_{x=x_{sep0}}}{\frac{dp}{ds}\bigg|_{x=x_{sep0}}} = C \frac{\partial w}{\partial s} \bigg|_{x=x_{sep0}}
\]

Eq. 75

where \( C = C(\lambda_{sep0}) \) gives the shift of the separation point due to the shift of the nozzle wall slope.
The aerodynamic pressure force per arc length due to a small wall displacement, may be written as:
\[ \ddot{F}_a'(\tilde{w}) = \dot{n}(p_a - p_{sep})(x_{sep} - x_{sep0}) = \dot{n}(p_a - p_{sep})C(x_{sep0})\frac{\partial w}{\partial s} \quad \text{Eq. 76} \]

By integrating the force contribution around the separation line the aerodynamic pressure force becomes:

\[ \bar{F}_a(\tilde{w}) = \int_{x_{sep}} \bar{F}_a' \, dy = (p_a - p_{sep})\int_{x_{sep}} \bar{n}C\frac{\partial w}{\partial s} \, dy \bigg|_{x_{sep}} \quad \text{Eq. 77} \]

The corresponding aerodynamic torque is:

\[ \bar{M}_a(\tilde{w}) = \int_{x_{sep}} \bar{x} \times \bar{F}_a' \, dy = (p_a - p_{sep})\int_{x_{sep}} \bar{x} \times \bar{n}C\frac{\partial w}{\partial s} \, dy \bigg|_{x_{sep}} \quad \text{Eq. 78} \]

The change of the nozzle wall slope at different azimuthal locations, \( \varphi \), due to a small tilt angle, \( \theta \), of the nozzle can be expressed as:

\[ \frac{\partial w}{\partial s} = \theta \sin \varphi \quad \text{Eq. 79} \]

Using this and

\[ \int_{x_{sep}} \ldots dy = \int_{0}^{2\pi} \ldots r(x_{sep}) \, d\varphi \quad \text{Eq. 80} \]

for small wall displacements, the aerodynamic torque can be expressed as:

\[ \bar{M}_a(\theta, x_{sep0}) = (0, M_a, 0) \quad \text{where} \]
\[ M_a(\theta, x_{sep0}) = (p_{sep} - p_a)Cr\pi(x\cos\tau + r\sin\tau)\theta \quad \text{at} \ x = x_{sep0} \quad \text{Eq. 81} \]

The eigenfrequency for the mechanical system alone is formed by inserting a harmonic amplitude ansatz:

\[ \theta \sim e^{i\omega t} \quad \text{Eq. 82} \]

into equation 61 and leaving out the aerodynamical torque \( M_a(\theta, P_{cc}) \):

\[ -J_y\omega^2\theta = -k\theta \quad \text{Eq. 83} \]

This gives:

\[ \omega^2 = \frac{k}{J_y} \quad \text{Eq. 84} \]

Now, consider the nozzle displaced when subjected to mechanical and aerodynamical loads and again assume the motion to be purely harmonic, i.e.:

\[ \theta \sim e^{i\Omega t} \quad \text{Eq. 85} \]

Introducing equation 84 and 85 in 61 and rearranging gives:
\[ \left( \frac{\Omega}{\omega} \right)^2 = 1 - \frac{M_a(\theta, P_c)}{k\theta} \]  

Eq. 86

Inspection of equation 86 shows that:

1. When \( M_a(\theta, P_c) < 0 \), the aeroelastic torque acts to restore the nozzle to its nominal position, i.e. the system becomes stiffer than the mechanical structure itself and the frequency of the eigenmode is shifted to a higher frequency, i.e. \( (\Omega/\omega)^2 > 1 \). This is for instance the case in a full flowing Vulcain nozzle.

2. When \( M_a(\theta, P_c) \in [0, k\theta] \), the aeroelastic torque acts in the same direction as the displacement of the nozzle wall, i.e. the system becomes weaker than the mechanical structure itself and the frequency of the eigenmode is shifted to a lower frequency, i.e. \( (\Omega/\omega)^2 \in [0, 1] \).

3. When \( M_a(\theta, P_c) |_{y^> k\theta} \), the unconditionally stable eigenmode becomes aeroelastically unstable, i.e. \( (\Omega/\omega)^2 < 0 \), and the displacement of the nozzle will thus start to grow exponentially.

The aeroelastic stability of the system can also be investigated for small displacements by substituting the linearised aerodynamic torque, equation 81, in equation 86, which gives:

\[ \left( \frac{\Omega}{\omega} \right)^2 = 1 - \left( \frac{p_{sep} - p_a}{C r \pi (x \cos \tau + r \sin \tau)} \right)_{y^< \text{sys}} \]  

Eq. 87

8.4.2 Experimental verification of the aeroelastic analysis

Within the GSTP FSC test programme, the influence of structural response and aeroelastic coupling were investigated for the VOLVO S1 model nozzle. This was done by varying the natural eigenfrequency of the bending mode with the use of exchangeable torsion springs. The mechanical eigenfrequencies of the bending mode are listed in Table 5 for the different spring set-ups used. The frequencies were determined by performing a ping test on the model set up, see section 7.3.1. A more detailed description of the test programme is presented in R 5.

<table>
<thead>
<tr>
<th>Spring name</th>
<th>Super weak</th>
<th>Weak</th>
<th>Medium</th>
<th>Stiff</th>
<th>Rigid</th>
</tr>
</thead>
<tbody>
<tr>
<td>Natural frequency</td>
<td>25.2</td>
<td>36.3</td>
<td>45.0</td>
<td>57.5</td>
<td>120</td>
</tr>
</tbody>
</table>

Table 5. Mechanical eigenfrequencies of the bending mode for the different spring set-ups with the VOLVO S1 nozzle, also published in Östlund et alR5, R108.

The aeroelastic stability of the S1 nozzle can be calculated from Eq.87 for the different spring set-ups. Such a calculation is presented in Figure 80 with \( p_{sep}/p_a = 0.3 \) and \( B' \) from an Euler calculation according to Figure 78. From the figure we can conclude that the only aeroelastic unstable system is the S1 nozzle with the super weak spring, \( (\Omega/\omega)^2 < 0 \) when \( x/L > 0.83 \), whereas the other systems are aeroelastic stable.

![Figure 80](image_url)
The aeroelastically stable system will almost behave like a regular forced response system, i.e. the closer the mechanical eigenfrequencies are to the frequencies of the aerodynamic load the higher generated loads. The only aeroelastic effect is that a small shift of the system eigenfrequency and a corresponding small amplification of the forced response load will occur. The frequency shift and the aeroelastic side-load amplification depend on the degree of coupling. For the weak, medium, stiff and rigid spring set-ups considered here the coupling is considered to be weak and the aeroelastic effect small.

For an aeroelastically unstable system, on the other hand, we would expect a significantly higher side-load magnitude compared to the classical forced response theory. When the separation enters the section of the nozzle that is unstable, the displacement of the nozzle will start to grow exponentially. At the same time the separation line will be displaced accordingly. The non-linear growth of the nozzle displacement will eventually saturate as parts of the separation line start to move out of the nozzle when the displacement becomes sufficiently large. This can be seen in the non-linear stability relation, equation 86, displayed in Figure 81 for the S1 nozzle with tilt angles $\theta=0.1^\circ$ and $\theta=2.6^\circ$. As the nozzle displacement saturates, the amplitude will eventually drop and the process will be repeated in a cyclic way.

Figure 81 shows that the aeroelastic instability occurs at a pressure ratio of $p_0/p_a=26-28$. When the pressure ratio is increased further, the nozzle will eventually become full flowing at $p_0/p_a=30$, and the system becomes stiffer than the mechanical structure itself, i.e. $(\Omega/\omega)^2>1$, since the aerodynamic torque now acts to stabilise the nozzle.

Let us now compare the model result with experimental data. For the side-load peak at $p_0/p_a=28$, Table 6, we can see that the test data correlate well with the aeroelastic predictions in Figure 80 and Figure 81. The aeroelastic system, i.e. super weak spring set-up, gives the highest loads ($M/M_{max}$) and interrupts the trend obtained for the aeroelastically stable systems. The systematic behavior of decreasing response load with decreasing spring stiffness for the rigid to the weak spring which are aeroelastic stable can be explained with classical forced response theory. The side-load peak at $p_0/p_a=28$ is caused by the separation shock system pulsating with a frequency of about 100 Hz at the exit. Forming the frequency ratio between the aerodynamical force and the mechanical systems, Table 4, we can see the above stated fact. The highest response load is obtained for the rigid spring, which has the frequency ratio closest to unity and the response
decreases with increased distance from the resonance value. The conclusion is thus that the aeroelastic theory is capable of predicting the obtained side-load features.

<table>
<thead>
<tr>
<th>Spring</th>
<th>( \frac{\omega}{\omega_n} )†</th>
<th>No. of tests</th>
<th>( M )</th>
<th>( M_{\text{max}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rigid</td>
<td>0.8</td>
<td>2</td>
<td>0.66</td>
<td></td>
</tr>
<tr>
<td>Stiff</td>
<td>1.7</td>
<td>10</td>
<td>0.63</td>
<td></td>
</tr>
<tr>
<td>Medium</td>
<td>2.2</td>
<td>10</td>
<td>0.48</td>
<td></td>
</tr>
<tr>
<td>Weak</td>
<td>2.8</td>
<td>10</td>
<td>0.45</td>
<td></td>
</tr>
<tr>
<td>Super Weak</td>
<td>3.9</td>
<td>5</td>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>

Table 6. Measured side-load magnitude versus frequency ratio between the exiting load and the mechanical system, peak at PR=28, also published in Östlund et al.²⁰⁸.

Finally, we compare the predictions of the aeroelastic stability for the S6 nozzle with experimental data, see Figure 82. The experimental frequency shift of the eigenmode has been determined from the strain signal. This was done by applying the Welch method²¹⁰ for power spectral analysis on the measured steady state side-load at different constant pressure ratios. The sampling time was at least 8 seconds for each case in order to achieve a sufficient frequency resolution. The linearised aeroelastic stability relation, equation 87, for the S6 nozzle has been calculated with \( \frac{p_{\text{sep}}}{p_{\text{a}}}=0.3 \) and \( B' \) extracted from an Euler calculation. In Figure 82 the linear stability relation based on small perturbation theory is also shown in order to visualise the effect of the over-prediction of the pressure shift this theory causes.

![Figure 82. Comparison between measured and calculated frequency shift for the S6 nozzle, also published in Östlund et al.²⁰⁸.](image)

In the separated flow region, the linearised stability theory predicts the same trend for the frequency shift as observed in experiments. The discrepancy is mainly due to the fact that both structure and gasdynamic damping are neglected in the model. If damping were included in the model the frequency shift would increase and the model prediction come closer to experimental data. However, the influence of the damping

\[ \omega_n = 100 \times 2 \pi \text{ [s}^{-1}\text{]} \] is the frequency of aerodynamical load, evaluated from Schlieren video and pressure measurements. \( \omega_n \) natural frequency of mechanical system.
is only significant during steady state operation whereas during short transient phases, such as a rocket engine start up, the damping plays a minor role and the model assumptions will thus come closer to reality. The increased system frequency observed in the experimental data when the nozzle is full flowing can not be captured with the linearised stability relation, i.e. Eq.87. However, as indicated in Figure 81, this effect can be captured with the non-linear relation given by Eq.86.

To summarise, we have seen that the modified aeroelastic model is capable of predicting the aeroelastic behaviour experienced in the tests. Further, the necessity of a case-sensitive method to determine the normalised pressure shift coefficient $B'$ has been pointed out.
Compressible gas flows lend themselves particularly well to optical methods of investigation. As a light ray travels through a compressible gas with density variations (which are related to variations in index of refraction), it undergoes three effects. First, a displacement from the path it would have taken in a uniform medium. Second, the angular displacement with respect to an undisturbed path, and third, a phase shift from the undisturbed light ray. These three effects correspond to the three main experimental visualisation techniques for compressible flow.

1. **Interferometry.** The fringe patterns in an interferogram arise due to the phase shift of light as it moves through a density field. This method is primarily suited for quantitative determination of the density field as it measures directly the changes of density.

2. **Schlieren imaging.** This experimental technique depends upon the change in the refractive index as a function of the density of the gas. In fact, schlieren imaging relies on the angular deflections of light rays. The intensity of the schlieren images corresponds to the gradient of the density, i.e. \( \nabla \rho \) or \( \nabla_x \rho \) in a 2-dimensional case depending on the test set-up. Although it is theoretically adaptable to quantitative use, it is inferior to the interferometer in this respect, and its greatest utility is in giving an easily interpretable picture of the flow field together with a rough picture of the density variations in the flow.

3. **Shadowgraphs.** This technique relies on the displacement of a light ray due to the change in refractive index because of spatial density variations in the gas. It can be shown that the displacement experienced by a light ray depends on the second derivative of the density, i.e. \( \nabla^2 \rho \). Therefore it makes visible only those parts of the flow where the density gradients change very rapidly, and it has found its greatest utility in the study of shock waves.

Of the three methods mentioned, the interferometer yields most information and the shadowgraph least. On the other hand, the interferometer is the most costly and the most difficult to operate, whereas the shadowgraph is the least costly and the easiest to operate. Each method therefore has its own useful niche in experimental work, and the choice of method depends on the nature of the investigation.

For supersonic, axially symmetric nozzle flows, these methods are only practically applicable to the exhaust flow. Typical results obtained with the schlieren method and with the shadowgraph method are shown in Figure 22, Figure 83 and Figure 84 respectively for a number of different sub-scale nozzles. To exemplify the usefulness of these methods in the case of highly overexpanded nozzle flows it can be mentioned that schlieren photos as the ones shown in Figure 22 have been an important source for the understanding of the physics behind the cap shock pattern. High-speed video recording of the schlieren pattern during transient operation was used extensively in the VOLVO S1-S8 campaigns, and has given valuable information of e.g. the transition from FSS to RSS and vice versa, see Figure 83. While schlieren photos at stationary conditions have been used to validate CFD results, see Figure 106.
Figure 83. Schlieren photos of the exhaust plume in the VOLVO S1 nozzle at different operational
conditions (from Östlund). 

Figure 83 shows schlieren photographs of the flow field at 9 different steady state operational points, 
$p_o/p_a = 10-50$ in steps of 5 for the VOLVO S1 nozzle. To support the understanding of the schlieren photos, cf. 
the corresponding calculated Mach number contours shown in Figure 61. From the schlieren photos we can 
see some typical flow features:

- At $p_o/p_a = 10$ no sharp density changes in the flow field can be detected and FSS is prevailing in the 
nozzle.
- At $p_o/p_a = 15$ the flow pattern has changed to RSS (the nozzle appears to be full flowing as the jet is 
attached to the wall) and the expansion and compression waves in the supersonic jet causing the stripy 
pattern can clearly be seen.
- Between $p_o/p_a = 25-30$ the reattachment point moves out of the nozzle.
- At $p_o/p_a = 40$ the mach disc has completely moved out of the nozzle.

Typical shadowgraph photos are shown in Figure 84 for two different CTIC nozzles at FSS condition tested 
at NAL. In the left photo the flow field is shown for the CTP86L nozzle at a low-pressure ratio, i.e. at 
significant overexpansion of the flow, and as can be seen the change of the density gradients is small and jet 
occupies only about 50% of the nozzle exit diameter. The flow field in the CTP50-R5-L nozzle can be found 
in the right photo. Due to less overexpansion, the CTP50-R5-L jet occupies a larger fraction of the nozzle 
exit diameter, ~75%, and the shadowgraph image is sharper compared to the CTP86L case.
Infrared radiometry (IR) is another experimental technique by which can contribute to the insight in the separation behaviour of nozzle flows. In recent experiments performed at DLR, IR has been used to visualise the wall temperature inside a sub-scale TOP nozzle operated at separated flow conditions. Typical results from this test campaign are shown in Figure 85-Figure 87 at FSS and RSS conditions respectively. It is clearly seen that a wall temperature increase is induced in the incipient flow region, in where the flow is still physically attached to the wall, see Figure 85 and Figure 86. In the case of reattachment of the jet to the wall, Figure 86, the wall temperature reaches a plateau value after this first temperature increase and then begins to decrease towards a constant value at the point where the flow reattaches the wall. In between the incipient separation line and the reattachment line a closed recirculating zone is established. Further downstream a second temperature peak can be observed, which is believed to be the affect of a second recirculation zone. IR-images have also made it possible to detect axial lines along the wall, which are believed to be caused by Görtler vortices. Figure 87 shows an image where such lines are clearly visible downstream of the throat. As seen in Figure 86, traces of such vortices, which originate in the upper part of the nozzle, can be found downstream of the flow separation line and it is believed that they may have an influence on the separation and wall heat transfer behaviour.
Figure 86. Infrared image: DLR TOP nozzle, pressure ratio $p_r/p_a=21.8$ during shut down, temperature range: 241-295 K, experiment by Stark et al. (Courtesy of DLR and ASTRIUM)

Figure 87. Infrared image: DLR TOP nozzle, vortex origin shortly after nozzle throat, experiment by Stark et al. (Courtesy of DLR and ASTRIUM)

In a paper by Hagemann et al., an illustrative figure is given on the transition process from FSS to RSS, see Figure 88. The figure shows pictures obtained by a regular Video recording of the nozzle wall. Here the separation and reattachment line can be identified from the onset of intensive radiation from the wall as the attached jet heats up the nozzle wall.
9.3 SEPARATION LINE VISUALISATION

The most basic common for detection of the separation line is by means of surface flow visualisation techniques consisting of a light coating of oil on the wall surface. In Figure 89 typical results obtained with this method are shown from the experimental study of a TOP nozzle by Girard and Alziary. They used a mixture of oil and carbon black to visualise the separation line. In order to visualise the reattachment line a much more viscous mixture of oil and grease was used. The mixtures were painted on the wall before each run downstream of the estimated separation and in a band over the estimated reattachment line respectively. During the run the oil and carbon black mixture would move upstream towards the separation line allowing a precise determination of its location. At the same time some of the oil and grease mixture would move downstream of the stagnation point and some would move upstream in the recirculation zone. Due to the low shear stress in the vicinity of the reattachment line, an amount of the mixture will remain around the reattachment line for several seconds.

Figure 89. Oil film visualisation of separation line in a TOP nozzle, experiment by Girard and Alziary. FSS line (left) and separation and reattachment line (right). (Courtesy of LEA Poitiers)
A novel method for separation line visualisation in internal flow is the use of Shear Sensitive Liquid Crystals (SSLC). The method consists of applying SSLC on the inner wall surface of a transparent specimen. As the colour of the crystals is a function of the shear stress, the separation line corresponding to $\tau_w=0$ is visualised. This technique has recently been applied to highly overexpanded nozzle flows by Tomita et al., see Figure 90-Figure 91. Figure 90 shows the SSLC pattern and the corresponding shadowgraph image, also shown in Figure 84, for two different compressed truncated ideal nozzle contours at free shock separation conditions. As can be seen in the figure a sharp and symmetric separation line is visible in both nozzles. The asymmetrical movement of the separation line and the sudden downstream shift of the separation position during transition from FSS to RSS are clearly visible in Figure 91.

Figure 90. Visualisation of separation line with Shear Sensitive Liquid Crystals (SSLC) and shock visualisation with shadowgraph in the CTP86L (left) and CTP50R5L (right) nozzle by Tomita et al. (Courtesy of NAL)

Figure 91. Two instant pictures of the wall shear stress field before (left) and after (right) transition from FSS to RSS in the CTP50R5L nozzle. (Courtesy of NAL).

9.4 FLOW VECTOR VISUALISATION OF EXHAUST PLUME FLOW

The simplest method to visualise the flow direction in an exhaust plume is to insert a wire with threads (tufts) into the flow. Figure 92 shows typical experimental results obtained by Stark et al. when applying this technique to a TOP nozzle together with a numerical calculation superimposed on the photo. The figure clearly shows the presence of a stable recirculating flow region in the plume by the threads, which are directed upwards. The indicator shows strong fluctuations in the recirculating zone downstream of the nozzle as well as in some outer regions. The movement of the threads is in good agreement with the calculated flow vector field, especially the location of the recirculating flow region at the centre line. The trapped vortex behind the cap shock pattern has been found in several CFD calculations, however, it has been questioned whether the trapped vortex is a numerical artefact or if it really exists in this type of flow. The experiment by Stark et al, Figure 92, is important, as it is the first to validate the existence of a
recirculation region behind the cap shock. At present more detailed analysis of this phenomenon using laser imaging techniques is carried out by various partners in the FSCD group.

Figure 92. Visualisation of flow field in the plume of the DLR TOP nozzle by using threads, experiment by Stark et al. (12). (Courtesy of DLR and ASTRIUM)
10 SCALING CONSIDERATIONS

Model experiments give valuable information for the understanding of physical phenomena in nozzle flow as well as for validation of models for separation and side-load prediction. The main advantages with scaled laboratory experiments compared to full scale testing are that:

- the boundary conditions are more well defined and easier to control and vary
- more sophisticated and accurate measurement techniques can be used and thus more detail information of the examined phenomenon can be extracted
- the model are cheaper to manufacture and instrument
- the cost for the testing is lower

When performing experimental studies in laboratory scale, scaling laws are required to define an experiment, which simulates the actual phenomenon in focus in the full-scale configuration. A complete matching of all parameters between sub-scale and full scale is not always possible, hence a priority must be made and as a consequence the similarity becomes restricted to the most significant parameters for the process studied. In this section we will give some basic ideas of the scaling considerations for the investigation of separation and side-loads in a real rocket nozzle, and in particular the Vulcain nozzle. A more general description of scaling laws for nozzle aerodynamic design can be found in the report by Koppenwallner R 106. This report treats similarity and scaling of inviscid and viscid nozzle flow of ideal gases based on stream tube approximation.

10.1 AERODYNAMIC SCALING OF NOZZLE FLOWS WITH IDENTICAL GASES

When gas (mixture ratio of the propellant) and operational condition (total pressure and total temperature) are the same as in the full-scale nozzle, a pure geometrical scaling can be used. In this case, the inviscid variables, such as Mach number, pressure, temperature etc. distributions, will be identical between the scaled nozzle and the original nozzle, i.e.

- \( M(x/L_{ref}, r/L_{ref})|_{scaled} = M(x/L_{ref}, r/L_{ref})|_{original} \)
- \( p/p_0(x/L_{ref}, r/L_{ref})|_{scaled} = p/p_0(x/L_{ref}, r/L_{ref})|_{original} \)
- \( T/T_0(x/L_{ref}, r/L_{ref})|_{scaled} = T/T_0(x/L_{ref}, r/L_{ref})|_{original} \)

where \( L_{ref} \) is a reference length, such as e.g. the nozzle length or throat radius.

Parameters containing length, such as Reynolds number and boundary thickness etc., on the other hand, will be reduced proportional to the geometric scaling factor \( s = L_{ref}|_{scaled}/L_{ref}|_{original} \). Flow separation is rather insensitive to Reynolds number as long as \( Re_\delta > 10^5 \) and it is hence important to make sure the subscale \( Re_\delta \) is in the same range as the full-scale nozzle. In the Vulcain engine the Reynolds number, based on the displacement thickness of the boundary layer, is of order \( 10^7 \), i.e. the Reynolds number is high and the flow is turbulent, thus the viscid flow features will not become significant different in a scaled nozzle with e.g. \( s=0.1 \). This type of sub-scale tests was performed during the development of the Vulcain engine. The test nozzle \( R^{107} \) in this case was a complete scale-down of the Vulcain nozzle. As expected the separation characteristics in the scaled nozzle \( R^{107} \) showed close agreement with the Vulcain nozzle \( R^{3} \). For instance, the transition of the separation pattern inside the nozzle from FSS to RSS and the transition from RSS to FSS at the exit of the nozzle occurred at the same operation conditions as in the Vulcain nozzle. However, the test and instrumentation cost for this kind of test is high. Further more, the test duration time is usually short due to test rig limitations, restricting the information level. It is therefore necessary to complement with subscale testing in wind tunnels, where testing and instrumentation are less expensive and the test duration time can be significantly increased.
10.2 AERODYNAMIC SCALING OF NOZZLE FLOWS WITH DIFFERENT GASES AND NOZZLE CONTOURS

When changing the working medium, as e.g. from the hot propellant gases in Vulcain with a specific heat ratio varying between γ=1.14-1.24 to air with γ=1.4, no simple scaling law as the one above exists. Using air in a wind tunnel model, a given pressure ratio or Mach number is reached at a smaller area ratio than for Vulcain, and an exact correspondence between the flows can thus not be achieved. Within the flow separation activities at VACR5, two different attempts were made to scale the Vulcain nozzle for sub-scale testing with air, which resulted in the VOLVO S1 and S3 model nozzles. The idea behind the scaling of these two nozzles and their resulting properties will be discussed below.

10.2.1 Reynolds number similarity

In general viscous and inertial length scales differs, and hence it is impossible to have both an inviscid and viscid similarity. Since the inviscid similarity is more important for the physics of flow separation, the Reynolds numbers was simply chosen to be of the same order of magnitude in the scaled nozzle as in the original nozzle. Based on a characteristic length y, e.g. axial distance or local nozzle diameter, the local Reynolds number along the nozzle can be written as:

\[ \text{Re}_{yi} = \frac{\rho V_i y}{\mu(T_i)} \]

The dynamic viscosity at \( T_1, \mu(T_1) \), can be related to \( \mu(T_0) \) by a viscosity law, e.g. the law of Sutherland or an exponential law. The local Reynolds number is thus related to the stagnation condition Reynolds number based on throat diameter,

\[ \text{Re}_{0d} = \frac{\rho_0 a_0 d_t}{\mu(T_0)} = \frac{p_0 d_t}{\sqrt{R_0 T_0} \mu(T_0)} \]

by

\[ \text{Re}_{yi} = \text{Re}_{0d} \cdot \frac{y}{d_t} f(M_1 \text{ and gas properties}) \]

The function \( f \) is independent of nozzle size and gives the distribution of the local Reynolds number in the nozzle as a function of Mach number and gas properties. Although an identical Mach number distribution can be achieved, the difference in gas properties, in particular \( \gamma \) makes it impossible to reproduce the Reynolds number distribution in the scaled nozzle. However, using the same stagnation Reynolds number, similar order of magnitude is achieved. A similar argument can be made concerning the similarity of Reynolds number based on boundary layer thickness.

10.2.2 Geometric and dynamic similarity

The effect of changing the working gas is easily demonstrated by the stream tube relations for calorically perfect gases. If we assume an identical Mach number distribution between the original and the scaled nozzle operated with gases with different \( \gamma \), the Mach number – area ratio formula (Eq. 1) shows that the area ratio must be different in the scaled nozzle:
With $\gamma_{sc}=1.2$ in the original nozzle and $\gamma_{sc}=1.4$ in the scaled nozzle will thus result in a nozzle with a smaller area ratio, as shown in Figure 93.

Figure 93. Ratio between scaled and original expansion ratio to keep identical Mach number distribution.

The above equation is only valid for scaling between nozzle flows with constant specific heat ratio. In Vulcain, the scaling relation is actually more complex, since the specific heat ratio varies in the range $\gamma=1.14-1.24$. Therefore a parametric study of model contours was done varying the values of the different contour variables $r_{sd}$, $\theta_{b}$, $L$, $r_{e}$, and $\theta_{e}$ (see section 3.5.1), in order to reach a final test nozzle contour that fulfilled the specified similarities.

Since the separation and side-load characteristics differ between different families of contours, the least requirement for geometric similarity is that the model nozzles must be of the same type as the original nozzle. Hence, in order to simulate the flow features in the Vulcain nozzle, the model nozzle should have the same parabolic type of contour as the Vulcain nozzle.

According to the Schmucker criteria\textsuperscript{94} the separation position is a function of the wall Mach number and the wall pressure, thus we need to achieve a similarity of these distributions in order to model the separation behaviour properly i.e.:

$$M_{wall\, model} = M_{wall\, Vulcain}$$

$$\frac{P_{wall\, model}}{p0} = \frac{P_{wall\, Vulcain}}{p0}$$

According to both the Schmucker side load model, see equation 47, and the aeroelastic side load model by Pekkari, see equation 75 and 87, the side load magnitude is further influenced by the local pressure gradient along the wall. The following similarity was therefore considered:

$$\frac{e(x/L_{ref\, sc})_{sc}}{e(x/L_{ref\, or})_{or.}} = \frac{2}{\gamma_{sc} + 1} \left(1 + \frac{\gamma_{sc} - 1}{2} M(x/L_{ref\, sc})^2\right)^{\frac{\gamma_{sc} + 1}{2\gamma_{sc} - 2}}$$

$$\frac{2}{\gamma_{or} + 1} \left(1 + \frac{\gamma_{or} - 1}{2} M(x/L_{ref\, or})^2\right)^{\frac{\gamma_{or} + 1}{2\gamma_{or} - 2}}$$
Apart from the flow properties along the wall, the internal flow field has a strong influence on the separation and side load characteristics in the nozzle, since the shape and strength of the internal shock emanating from the inflection point of the contour affects FSS to RSS transition. Therefore the entire flow field needs to be correctly modelled, in particular the Mach number distribution i.e.:

$$
\frac{dp_{\text{wall}}}{dx} \frac{L_{\text{ref}}}{p_0} = \frac{dp_{\text{wall}}}{dx} \frac{L_{\text{ref}}}{p_0}_{\text{Vulcain}}
$$

The choice of a characteristic length, $L_{\text{ref}}$, is not obvious. The prime candidates are the nozzle exit radius, $r_e$, and the nozzle throat radius, $r_t$. Using $L_{\text{ref}}=r_e$ the model contour will become closer to the original nozzle contour at the nozzle exit, whereas $L_{\text{ref}}=r_t$ results in longer and thus a more slender contour, and the contour similarity will be restricted to the throat region.

10.3 S1 AND S3: TWO WAYS OF SCALING DOWN THE VULCAIN NOZZLE FLOW

Based on the similarity considerations above, a parametric study of different TOP nozzles simulating Vulcain was performed. The resulting dimensions of model nozzles S1 and S3 are given in Table 7 together with the Vulcain dimensions. In Vulcain ($\varepsilon=45$) the averaged exit Mach number is $M_e=5$ and the nozzle length is $L_e=15.7r_t=2.3r_e$. For a nozzle operated with air, Mach 5 is reached with an area ratio of 25 according to stream tube relations, see equation 1. If the exit radius is used as the characteristic length, the length of the model nozzle will then be $L_e=2.3r_e=2.3\sqrt{25} r_t=11.5 r_t$, while the corresponding length when using the throat radius is $L_e=15.7r_t$, i.e. a much longer and a more slender contour is obtained. The chosen characteristic length, $L_{\text{ref}}$, was the nozzle exit radius, $r_e$, in sub-scale nozzle S1 and the nozzle throat radius, $r_t$, in sub-scale nozzle S3. In both cases, similarities of properties along the wall were considered. In addition to this, similarity in the overall flow field and shape of the internal shock was considered in the S3 nozzle.

### Table 7. Main characteristics of the different nozzles.

<table>
<thead>
<tr>
<th>Nozzle</th>
<th>Vulcain</th>
<th>S1</th>
<th>S3</th>
<th>Dimension</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter</td>
<td>Value</td>
<td>Value</td>
<td>Value</td>
<td></td>
</tr>
<tr>
<td>Area ratio ($\varepsilon$)</td>
<td>45</td>
<td>20</td>
<td>18.2</td>
<td>-</td>
</tr>
<tr>
<td>Nozzle length ($L$)</td>
<td>2065.5</td>
<td>350</td>
<td>528.2</td>
<td>mm</td>
</tr>
<tr>
<td>Throat diameter ($D_t$)</td>
<td>262.4</td>
<td>67.08</td>
<td>67.08</td>
<td>mm</td>
</tr>
<tr>
<td>Normalised inlet wall radius ($r_{iw}/r_t$)</td>
<td>0.5</td>
<td>0.5</td>
<td>3.0</td>
<td>-</td>
</tr>
<tr>
<td>Throat wall angle ($\theta_N$)</td>
<td>35.025</td>
<td>35.025</td>
<td>27.000</td>
<td>°</td>
</tr>
<tr>
<td>Nozzle exit angle ($\theta_e$)</td>
<td>6.5</td>
<td>4.0</td>
<td>0.0</td>
<td>°</td>
</tr>
<tr>
<td>Nozzle exit diameter ($D_E$)</td>
<td>1760.2</td>
<td>300.0</td>
<td>286.5</td>
<td>mm</td>
</tr>
<tr>
<td>Design feeding pressure ($p_0$)</td>
<td>11.0</td>
<td>5.0</td>
<td>5.0</td>
<td>MPa</td>
</tr>
<tr>
<td>Design feeding temperature ($T_0$)</td>
<td>3500</td>
<td>450</td>
<td>450</td>
<td>K</td>
</tr>
<tr>
<td>Feeding gas</td>
<td>LOX/LH2</td>
<td>Air</td>
<td>Air</td>
<td>-</td>
</tr>
</tbody>
</table>
Figure 94. Comparison of wall properties between Vulcain and model nozzle S1 and S3.
Figure 94-Figure 99, compares the distribution of different geometrical and dynamic quantities. As anticipated, see Figure 94, the S1 nozzle has the same bell like shape as Vulcain, whereas S3 is much more slender. The wall Mach number, wall pressure and pressure gradient of both model nozzles are similar to those in the Vulcain nozzle. We can also see the effect of different initial expansion regions. In S1, the same values of $\theta_i$ and $r_{th}/r_i$ were used as in Vulcain. This result in a higher wall Mach number at the end of the initial expansion compared to Vulcain, which is an effect of the higher value of $\gamma$ in air$^\ast$. In S3, $\theta_i$ was reduced, and as a consequence the wall Mach number is somewhat reduced at the end of the initial expansion region compared to the S1.

Comparing the internal Mach number distribution in the different nozzles, shown in Figure 95-Figure 97, we can see that the higher initial expansion in the S1 nozzle together with the relatively short nozzle length cause the kernel to occupy about 70% of the local nozzle cross section at all streamwise positions. As a consequence, the internal shock emanating from the contour inflection point is located closer to the wall compared to Vulcain. The higher wall Mach number at the inflection point and the more drastic turning of the flow will further result in a stronger internal shock in S1 compared to the S3 and Vulcain. Comparing the S3 and Vulcain nozzle flow, it can be seen that the similarity of the kernel flow is better. The radial extension of the kernel is gradually reduced from about 70-80% of local nozzle radius near the throat region to about 30% at the nozzle exit. Note also that the curvature of the Mach number contours is more similar to Vulcain in S4 than in S1, affecting the shock and separation pattern.

---

** With increasing $\gamma$ a smaller expansion angle is required to reach a specified Mach number as can be seen from the Prandtl-Mayer function.
Figure 94 shows the Reynolds number based on local nozzle diameter, $Re_d$. It can be seen that $Re_d$ increases monotonically along the wall of both S1 and S3, while it is approximately constant in the Vulcain. As discussed above, this is a problem which is inherent in the scaling of nozzle flows with different gas properties. Moreover, despite there small size, the sub-scale nozzles have a much larger $Re_d$ than Vulcain due to the higher stagnation density for air at reasonable values of the stagnation temperature compared with the hot propellant gases. In the test nozzles the value of the design stagnation temperature was 500 K in order to avoid condensation of the nitrogen when expanding the gas to high Mach numbers and this value was also the limit of the rig where the tests were performed. For a complete matching of the stagnation Reynolds number, the throat radius of the model nozzles would have to be about 0.01 m, however for instrumentation purposes a larger scale size, $r_t=0.03$ m, was chosen. The larger Reynolds number is expected to affect the separation length, $L_s/\delta$, with less than 10 percent $^{11}$. The effect of the difference in Reynolds number between the Vulcain and the model nozzles can thus be considered to be small.

The Reynolds number based on the displacement thickness, $Re_\delta$, is shown in Figure 98. The higher levels of $Re_\delta$ for S3 compared with S1 is due to the thicker boundary layer obtained in the much longer S3 nozzle, see Figure 99.

![Figure 98. Reynolds number based on displacement thickness in the Vulcain, S1 and S3 nozzle.](image1)

![Figure 99. Displacement thickness normalised with wall radius vs. axial distance in the Vulcain, S1 and S3 nozzle.](image2)

$^{11}$ According to experiments by Spaid and Frishett$^{54}$, see Figure 29a, a variation of $Re_\delta$ by a factor of 2 results in an increase of $L_s/\delta$ with less than 10% at a shock angle of $14^\circ$ and $Re_\delta \approx 10^4$. At $Re_\delta > 10^5$, $L_s/\delta$ is almost constant according to the experiments by Settles$^{50}$, see Figure 29b.
10.3.1 Experimental verification of the scaling

Figure 100 shows the shock pattern obtained for the model nozzles and Vulcain. As can be seen, the similarity to Vulcain regarding the shock patterns, especially the size of the normal shock, in the over-expanded jet is closer in the S3 nozzle compared to the S1 nozzle. This reflects the good similarity in the overall flow field and shape of the internal shock between S3 and Vulcain.

Figure 100. Exhaust plume patterns for the different nozzles, a) Schlieren photo of S1 VAC FFA, b) Schlieren photo of S3 VAC FFA, c) Calculated Mach number field (from blue to red) in the Vulcain nozzle.

Table 8 gives a comparison between the model nozzles and Vulcain regarding separation, side-loads and transition between separation patterns. We recall that S1 was designed with the exit radius as reference length while the throat radius was used for S3. The global separation and side load characteristics in the scaled nozzles, S1 and S3, show close agreement with Vulcain. Both S1 and S3 have the same type of transition phenomenon as Vulcain, one from FSS to RSS inside the nozzle and a second from RSS to FSS at the nozzle exit. These transitions occur almost at the same thrust levels in the model nozzles as in Vulcain. However, when it comes to the location of the incipient separation, \( x_i \), which occurs at the FSS condition before the transition, a large difference between the two models and Vulcain is obtained. The table shows that in the model nozzles \( x_i \) is located about 30% upstream of corresponding location in Vulcain. One reason is that the pressure recovery in the recirculating zone at FSS is sensitive to the downstream contour geometry, see Figure 46. This is the case especially in the S3 nozzle, which has a contour shape that looks quite different from Vulcain. In the S1 nozzle the discrepancy is rather due to an earlier transition compared with Vulcain, which is in turn an effect of the difference in the internal flow field between the nozzles. The pressure recovery in RSS on the other hand may be assumed to be less geometry-dependent due to the reattachment and subsequent shock/expansion system.

<table>
<thead>
<tr>
<th>Model</th>
<th>( x_i/L_{ref} )</th>
<th>( P_{0,\text{transition}}/P_{0,\text{nom}} )</th>
<th>( M_{SL}/L_{ref} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>S1</td>
<td>0.45</td>
<td>0.67</td>
<td>(0.66)</td>
</tr>
<tr>
<td>S3</td>
<td>0.72</td>
<td>(1.13)</td>
<td>0.88</td>
</tr>
<tr>
<td>Vulcain</td>
<td>1*</td>
<td>1*</td>
<td>1*</td>
</tr>
</tbody>
</table>

Table 8. Comparison of measured quantities between model nozzles and Vulcain. (*=from CFD calculations, \( x_i \)=incipient separation point, \( L_{ref} \)=reference length, \( r \)=radius, \( t \)=throat, \( e \)=exit, \( P_{0,\text{nom}} \)=nominal stagnation pressure see Table 7, \( M_{SL} \)=side-load tourqe at FSS→RSS transition, values normalised with irrelevant \( L_{ref} \) are included within brackets for information)
Table 8 shows that the side-load level (MSL) is about 50% larger in S1 compared to Vulcain, while it is 35% lower than Vulcain in S3. This discrepancy is due to the large difference in contour geometry obtained when scaling Vulcain flow to a nozzle operated with a different gas (γ). This makes the relation between $x_i$ and $r_i$ different in the model nozzles compared to Vulcain, and thus scaling of quantities involving different length parameters can not be expected to be successful, such as the side-load. The sensitivity of MSL to different length scales can be understood by considering the following expression of the FSS to RSS transition side-load (cf. Section 8.1):

$$M_{SL} = \int_0^{L_{r_2}} \int_0^{\pi} x \cdot \left( p_w - p_a \right) \eta_\perp r d\phi dx = \int_0^{L_{r_2}} \int_0^{\pi} x \cdot \left( p_w - p_a \right) \eta_\perp r d\phi dx + \int_0^{L_{r_2}} \int_0^{\pi} x \cdot \left( p_w - p_a \right) \eta_\perp r d\phi dx$$

The first term is associated with the sudden downstream shift of the separation line during the FSS to RSS transition. The second term is the side-load contribution due to the difference in the wall pressure distribution between the separation patterns downstream of $x=x_{i,RSS}$. Neglecting the second term and approximating the first integral relation by:

$$\int_0^{L_{r_2}} \int_0^{\pi} x \cdot \left( p_w - p_a \right) \eta_\perp r d\phi dx \approx \left( x_{i,RSS}^2 - x_{i,FSS}^2 \right) \left( r_{i,FSS} + r_{i,RSS} \right) \Delta \rho$$

we finally get:

$$M_{SL} \propto \left( x_{i,RSS}^2 - x_{i,FSS}^2 \right) \left( r_{i,FSS} + r_{i,RSS} \right)$$

This gives

$$M_{SL}/L_{ref}^3 \bigg|_{Model} \approx \left( \frac{x_{i,RSS}^2 - x_{i,FSS}^2}{L_{ref}^2} \right)_{Model} \left( \frac{r_{i,FSS} + r_{i,RSS}}{L_{ref}} \right)_{Model} = C_1 \cdot C_2$$

and with values inserted we obtain:

$$\frac{M_{SL}/r_e^3}{M_{SL}/r_e^3} \bigg|_{S1} = 1.54 \cdot 0.95 = 1.5$$

and

$$\frac{M_{SL}/r_e^3}{M_{SL}/r_e^3} \bigg|_{Valcain} = 1.04 \cdot 0.57 = 0.6$$

which corresponds well to the scaled values of measured side-load, $M_{SL}$, given in Table 8. This also demonstrates that the side-load model described in section 8.1 is basically correct. We can also see that the source of the discrepancy is different for the model nozzles. In the S1 nozzle it emerges from a bad similarity of the separation locations, whereas in the S3 nozzle it comes from lack of similarity in geometrical proportions. One may therefore conclude that it is necessary to use two different length scales instead of only one. As seen in Figure 101, the exit radius, $r_e$, would be a better choice for the scaling of the nozzle radius.
Figure 101. Influence of different reference lengths on the scaling of nozzle radius between the S3 and Vulcain nozzle (top: $L_{ref} = r_t$ and bottom: $L_{ref} = r_e$).

When scaling axial coordinates with $r_t$ and radius with $r_e$ in the simplified expression for the side-load, i.e.:

$$
\frac{M_{sl}}{(r_t^2 r_e)} \left( \frac{x_{i, RSS}^2 - x_{i, FSS}^2}{r_t^2} \right)_{\text{Model}} = \left( \frac{r_{i, FSS} + r_{i, RSS}}{r_e} \right)_{\text{Model}} = \left\{ \begin{array}{l}
(0.69 \cdot 0.95 = 0.64 \text{ for S1}) \\
1.04 \cdot 0.89 = 0.92 \text{ for S3}
\end{array} \right.
$$

we obtain a much better agreement between S3 and Vulcain. The agreement is even more striking when applying the mixed scaling is applied to the measured side-load:

$$
\frac{M_{sl}}{(r_t^2 r_e)}_\text{Vulcain} = \left( \frac{r_{i, FSS} + r_{i, RSS}}{r_e} \right)_\text{Vulcain} = \left\{ \begin{array}{l}
(0.65 \text{ for S1}) \\
1.01 \text{ for S3}
\end{array} \right.
$$

The results and analysis shown above verify that the used methodology for scaling Vulcain to a sub-scale model operated with air has been successful. Both S1 and S3 have captured the relevant physical phenomenon found in Vulcain. The results from the model tests have been the basis for the understanding of the separation and side-load characteristics. They have, for the first time revealed the transition between the different separation patterns as a basic mechanism behind for side-load generation. It must be kept in mind that the key for a good similarity between a sub-scale and full-scale nozzle is still the design of a representative sub-scale nozzle contour. The S3 nozzle shows a good similarity to Vulcain, because the overall flow field and shape of the internal shock were considered in addition to flow properties along the wall. We have also shown that if this is done, the side-load moment is accurately reproduced by combining the two length scales as $r_t^2 r_e$. The S1 nozzle, however, were only the wall properties were taken into account, is only capable of reproducing the basic separation and side-load phenomena of Vulcain. Attempts to scale the model test results to Vulcain show that no simple generalised relation for scaling of separation location and side-load exists. The scaled results with S3 show good agreement whereas the scaled results with S1 show a large discrepancy. However, the detailed information of the different phenomenon obtained in the model tests has made it possible to develop generalised mathematical descriptions of the processes. With these analytical models the separation location, the transition between FSS and RSS and the corresponding side-load in rocket nozzles very different from the S1, S3 and Vulcain nozzle can be accurately predicted.
11 REYNOLDS-AVERAGED NAVIER-STOKES CALCULATION METHODS

The most common approach for predicting turbulent shock wave boundary layer interactions, including those involving separation, is to solve the Reynolds-averaged Navier-Stokes (RANS) equations. In the following a few problems specific to shock boundary layer interactions and flow separation will be addressed.

11.1 INTERACTIONS IN BASIC CONFIGURATIONS

A critical survey of current numerical prediction capabilities for simulation of laminar and turbulent interactions in basic configurations, such as the single fin, double fin and hollow cylinder flare, were presented by Knight and Degrez in 1998. The objective of their study was to determine how well current codes could predict quantities needed in the design of high speed vehicles, including flowfield structure, and mean and fluctuating aerodynamic and thermal loads. They concluded that for laminar flows existing codes accurately predict both aerodynamic and thermal loads. However, the situation for turbulent flows is not as satisfactory. They concluded that mean pressure distribution in 3-D interactions can be computed quite well, with little variation between computations using different turbulence models. On the other hand, skin friction and heat transfer distributions are generally poor, except for weak interactions (no separation), with different turbulence models producing different results. The differences between measured and predicted heat transfer are substantial. Knight and Degreze note differences up to 100% for strong interactions (separated flow). In 2-D interactions, especially strong ones, the situation is somewhat bleaker. Mean surface pressure distributions are satisfactory only for weak interactions. In strong interactions, the models generally predict too little upstream influence, i.e. the calculated separation length is shorter compared with the one observed in experimental data.

11.2 REALIZABILITY CONSTRAINTS

Much effort has been spent by different researchers on corrections that cure some of the apparent anomalies in RANS simulations of strong interactions. The most common corrections for compressible boundary layers are the realizability, the turbulent length scale limit and the compressibility correction. The first of these is described below.

The mathematical concept of realizability is that the variance of the fluctuating velocity components must be positive and the cross-correlations bounded by the Schwartz inequality. Solutions obtained for strong interactions with common two-equation turbulence models violate this realizability constraint in the outer part of the boundary layer and outside. The size of the unrealizable zones increases with the interaction strength and they are clearly related to the largest values of the dimensionless strain rate invariant in the flow, especially across the shocks.

A recent review of realizability correction of two-equation turbulence models recalls that the idea is to enforce the realizability constraints by limiting the value of the constant in the definition of the eddy-viscosity $\mu = \alpha C_{\mu} \rho k/\omega$ (where $\omega = \epsilon/k$) as follows:

$$\alpha_c = \min \{1, \alpha_c^* \}$$

where $\alpha_c^*$ is defined by:

$$\frac{1}{\alpha_c^* C_{\mu}} = A_0 + A_1 (s^2 + A_2 \sigma^2)^{1/4}$$

$C_{\mu}$ is the usual constant 0.09, $s$ is the dimensionless mean strain rate $S/\omega$ with $S^2 = 2S_{ij}S_{ij} - \frac{2}{3} S_{ij}^2$, and $\sigma$ is the dimensionless vorticity invariant $\sqrt{2\Omega_{ij} \Omega_{ij} / \omega}$ where:
Moore and Moore\textsuperscript{R 144} propose a set of constants (i.e. $A_0$, $A_s$, and $A_r$) derived from an Algebraic Reynolds Stress Model ($A_0=2.85$, $A_s=1.77$) and assumed, in a first approximation, that the strain rate and rotation have symmetrical effects ($A_r=0$). They show that, in the case of flows near leading edges, where the inviscid strain rate is very large ($s=100-400$), the modifications ends up with a much better prediction of the level of the turbulent kinetic energy ($k_t$). Other researchers have proposed other values of the constants, e.g. Durbin\textsuperscript{R 145, R 146} proposed a similar correction with $A_0=A_r=0$ and $1/A_s=0.29$. This gives smaller values of $\alpha$ then those obtained by Moore and Moore, but and is virtually identical to those by Coakley\textsuperscript{R 147, R 148} or by Menter\textsuperscript{R 149} using the SST model, where $1/A_s=0.3$.

The effect of this type of correction is illustrated in Figure 102, showing the pressure, Mach number, turbulent kinetic energy ($k_t$) and dissipation ($\omega$) distribution in a quasi one-dimensional nozzle, which adapts to the exit condition through a normal shock. The turbulence model used is the Wilcox standard $k$-$\omega$ model, with and without a realizability correction similar to the one proposed by Moore and Moore. Here after we label the standard model without correction as $WI$ and the one with correction as $WM$ respectively. The pressure and Mach number distribution obtained with the Euler equations are also included in the figure for comparison. As can be seen in the figure, the $WI$ model gives an unphysical increase of $k_t$ and $\omega$ already in the convergent part of the nozzle, where the flow is accelerated to sonic conditions. In a real case, strong acceleration can lead to relaminarisation of the flow and this trend is captured with the $WM$ model. With the $WI$ model the production of $k_t$ explodes over the shock, which smears out the shock and affects its predicted position. The $WM$ model cures this stagnation point anomaly at the normal shock. With this type of correction the results are generally improved, however, the results are still not satisfactory as will be illustrated in the following example of a nozzle at overexpanded flow conditions.

Figure 102. Influence of realizability corrections on a normal shock in a quasi one-dimensional nozzle

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad \text{and} \quad \Omega_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right).
\]
11.3 OVEREXPANDED NOZZLE FLOW

11.3.1 Geometry and Mesh

The nozzle studied is the VOLVO S6 nozzle tested in the HYP 500 windtunnel at FOI. The nozzle is an ideal truncated nozzle with design Mach number $M_D=5.15$ truncated at $\varepsilon=20.6$. The two dimensional axis-symmetric flowfield is discretised by 305x97 cells inside the nozzle with a $y^+$ value of the first wall cell lower than 1. Outside the nozzle about the same amount of cells are spent additionally. The extension of the ambient region in axial direction is around 85 throat radii and 75 in the radial direction. The grid distribution in the axial direction is increased in two zones, one over the separation shock and the other over the Mach disc. The position of these regions is adjusted to different operation conditions, as the separation shock and the Mach disc moves further downstream when the pressure ratio $p_0/p_a$ is increased. A typical grid is shown in Figure 103 for an operational condition where the separation shock and the Mach disc are both located near the nozzle exit region. The solutions were checked for grid independence for all operational conditions examined.

![Figure 103. Grid distribution in VolvoS6, the whole mesh domain (left) and a close up of nozzle exit lip (right).](image)

11.3.2 Computational model

The Volvo finite volume multi block structured grid RANS code VolSol is used to obtain steady state solutions. The numerical method used is an explicit third order Runge-Kutta time marching method using a local time-step. The mean flow convective flux is calculated with a third order accurate upwind-biased scheme. The same scheme with a TVD limiter is used for the turbulent convective flux. Viscous terms are discretized with a compact second order scheme. The effects of turbulence are incorporated through the eddy viscosity assumption and the turbulent Prandtl number $Pr_t=0.9$. The turbulence model used to determine the turbulent eddy viscosity is the Menter $RST$ two-equation model. The working fluid is air, modelled as a calorically perfect gas with the gas constant $R=287.2 \, J/(kgK)$ and an isentropic coefficient of $\gamma=1.4$ and Prandtl number $Pr=0.72$. The molecular viscosity is computed from Sutherland's law for air. A pressure boundary condition is used for all inflow and outflow boundaries, and a condition of $T_s=288 \, K$ and $p_s=1 \, \text{bar}$ is applied at the ambient boundaries. The stagnation conditions prescribed at the nozzle inlet were the ones obtained at the different operational conditions in the tests, i.e. $T_0=450-500 \, K$ and $p_0=10-70 \, \text{bar}$. An adiabatic wall with a low Reynolds formulation is applied to the internal nozzle wall. At the external wall surfaces of the nozzle slip conditions (Euler walls) are applied. The computations were performed at Volvo Aero Corporation and more details of the numerical method and this specific simulation can be found in reference R 150-R 152.

11.3.3 Comparison of computations with experiment

Figure 104 shows calculated and measured wall pressure profiles are shown for three different operational conditions, $p_0/p_a=10$, 20 and 70 respectively. As can be seen, the predicted incipient separation point, i.e. the first deviation from the pressure profile obtained with attached flow condition, occurs upstream of the
measured one for all operation conditions. The incipient separation position predicted for the $p_0/p_a=10$ case is closest to the experimental data and as $p_0/p_a$ increases the discrepancy increases, to finally become significant for the $p_0/p_a=70$ case. In all cases the predicted separation length is shorter than observed in the test data, which gives a steeper pressure rise in the separation region compared with experimental values.

The misprediction of the location of incipient separation point at high-pressure ratios also influences the predicted position of the Mach disc. Figure 105 shows the calculated Mach number distribution at $p_0/p_a=55$ and in Figure 106 the predicted shock system is compared with a schlieren image obtained for VOLVO S6 at the same operational condition. It can be seen that the Mach disc obtained in the simulation is located too far upstream compared with the test data. It is not clear if it is only the separation line that drives the location of the Mach disc or if other factors are involved.

![Figure 104. Wall pressure in the VOLVO S6 nozzle, comparison between Menter SST and test data.](image)

![Figure 105. Mach number distribution in the VOLVO S6 nozzle at $p_0/p_a=55$.](image)
The coefficients in the realizability constraint used in the Menter SST model is $A_0=A_r=0$ and $A_s=10/3$. Since researchers have proposed different values, it was necessary to assess the influence of $A_s$ on the computed flow field. Such a study was performed, where $A_s$ was varied within the range $1/3-10/3$, and lead to the conclusions that when $A_s$ is reduced the incipient separation point and corresponding shock system is moved further downstream and the opposite happens if it is increased. Thus by adjusting $A_s$, a better prediction can be obtained at high-pressure ratios, however, this will instead cause increased discrepancies at low pressure ratios. On the whole, the procedure of adjusting turbulence model parameters “on hand” is not satisfying, since it is somewhat arbitrary and does not guarantee adequate results for new types of flow fields.

It is clear from the above that the current capability to predict critical quantities for design of applications featuring strong shock wave boundary layer interactions is not satisfactory. One of the drawbacks is that the eddy-viscosity models use a single length scale to represent the turbulence, which is insufficient in separated flow. Secondly RANS calculations do not model flowfield unsteadiness. As noted earlier (see section 5.2.4 and especially Figure 34) the global flowfield unsteadiness is such a dominant feature in this type of flow, and that without modelling it, not even mean quantities can be computed accurately.

To address these deficiencies Knight and Degrez recommend the development of Large Eddy Simulation (LES) solvers. Unlike Direct Numerical Simulation (DNS) in which all scales of motion that contains significant energy are resolved. LES attempts “to resolve the eddies that are large enough to contain information about the geometry and dynamics of the specific problem under investigation and to regard all structures on a smaller scale as universal following the viewpoint of Kolmogorov” (Ghosal, 1999). In that sense LES occupy the middle ground between RANS methods (which are relatively cheap computationally) and DNS, which is presently prohibitively expensive, if not impossible, for most aerospace applications. However, from an engineering perspective, it is questionable whether LES or its variants will ever become design tools. Jameson notes that it is unlikely that designers will ever “need to know the details of the eddies in the boundary layer”. But he also argues that it is possible that LES may provide “an improved insight into the physics of turbulent flow, which may in turn lead to the development of more comprehensive and reliable turbulence models”, which in turn would improve RANS based modelling. From the recent literature it is evident that some very interesting work is now being done in LES and variants of LES. Additionally there is reason to hope that innovative approaches will evolve, which will solve existing problems and address the new ones, which will undoubtedly arise as LES is applied to higher Mach number flows.
12 SUMMARY AND CONCLUSIONS

The concern of the present work is to model flow separation and separation-induced side-loads in rocket engine nozzles. The aim is to prevent flow separation at design condition, or to predict the position of flow separation and the ensuing side-loads at overexpanded flow conditions. Different types of traditional bell-shaped nozzles exist, each producing their own specific internal flow field. The TIC nozzle is the only nozzle type that produces a shock free flow, whereas with other nozzle contours, such as the conical, CTIC, TOC, TOP or directly optimised nozzles, an internal shock is formed inside the nozzle as shown in Figure 3. The contour type also determines the flow pattern that can be observed in the exhaust plume. Basic patterns that can be observed are the classical Mach disk, the apparent regular reflection, and the cap-shock pattern, which are all shown in Figure 21-Figure 22. The latter is only observed for nozzles featuring an internal shock that exits the nozzle and is then reflected at the centreline. In a nozzle operating with separated flow, there can be two different separation patterns, the free shock separation (FSS) and restricted shock separation (RSS), see Figure 37-Figure 38. The cap shock pattern is a driver for a transition of the flow from FSS to RSS, and hence the RSS pattern exists only in nozzles that produce a cap-shock. Transitions between FSS and RSS and vice versa are the origin of side-load peaks, characterised by their high level and impulsive occurrence. As a consequence, the separation and side-load characteristics are radically different for different contour families, and hence the choice of contour type an essential for the mechanical load picture. This is of particular importance for first-stage nozzles, since they are started and partly operated at high ambient pressure. For upper-stage engines, the choice of contour is of minor importance, since in that case there is no considerable flow separation at start up, unless they are used for stage separation.

12.1 NOZZLE FLOWS WITH FREE SHOCK SEPARATION (FSS)

12.1.1 Flow separation prediction

All nozzle types display FSS as long as the pressure difference between the combustion chamber and the ambient is still low and the separation is located well inside the nozzle. In this case, the separation position is predicted on the basis of models for the pressure rise from attached flow wall pressure to ambient pressure. Various simplified models of this kind have been published in literature, and show qualitative agreement with experimental data. However, to achieve a higher accuracy in quantitative prediction, two different physical phenomena associated must be decoupled, namely (i) the pressure increase due to the shock wave boundary layer interaction, and (ii) the pressure recovery associated with the recirculating flow in the separated region.

For the first of these, the shock wave boundary layer interaction phenomenon, the present thesis suggests a new criterion based on the generalised free interaction theory by Carrière et al. With this criterion the start of the interaction region as well as the pressure distribution and the corresponding length of the intermittent zone can be predicted in a nozzle at a given operation condition and a prescribed plateau pressure. A correct value of the interaction length is essential, since it is one of the main elements in FSS side-load models. The results look encouraging when comparing to sub-scale test data, however more effort are needed before a reliable and accurate criterion can be established for real rocket nozzles. Especially, the applicability to chemically reacting flow cases, where the value of the specific heat ratio is different compared to air, needs to be validated. The influence of wall cooling is another topic, which needs to be investigated in detail. VOLVO, ASTRIUM and DLR are currently preparing a test campaign in order to shed light on these open ends. Further, this criterion must be coupled with an accurate model describing the pressure rise in the recirculating zone. Today, a constant value of the plateau pressure is often used, based on test data experience. This approach works quite well for flow separation far from the nozzle exit whereas the discrepancy increases with decreasing distance to the nozzle exit. Thus, for accurate prediction of the separation location, models for the pressure rise in the recirculating zone in contoured nozzles operated with hot propellants needs to be developed. A possible semi-empirical approach to construct such a model on is Abramovich’s theory for the mixing of counterflowing compressible turbulent jets.
12.1.2 Side-load prediction

Experiments on basic interactions have shown that the shock wave boundary layer interaction is an intermittent and 3-dimensional phenomenon. Mechanical structures exposed to this type of supersonic flow separation will be affected by large, time-dependent forces, which can be resolved into two components, a low frequency buffeting caused by changes in the geometry of the separation region, and high frequency fluctuations originating from the shear layer of the separated region. In the Schmucker model these geometry changes of the separation region are modelled in a quasi-static manner as a tilted separation line, i.e. as asymmetric pressure distribution acting over an effective area. The logic of the Schmucker model construction is in that sense correct. However, the physical treatments of some elements in the model are incorrect, e.g. the interaction length, which has no coupling to the incoming boundary layer properties at all and is not influenced by changes of the geometry downstream of the separation region. Apart from the Schmucker model, a number of other side-load models based on the tilted separation line approach can be found in the literature, which differ from the Schmucker model regarding both physical and logical bases, however their usefulness is questionable.

The oscillation of the separation line and pressure pulsation in the separated flow region is also the basis of the Dumnov side-load model. In contrast to Schmucker, however, Dumnov uses experimentally determined wall pressure fluctuations and their inter-correlation to construct a generalised pressure fluctuation function from which the side-load level is estimated. According to Dumnov, the accuracy of the model is within 20%. Magnitude of the experimentally determined pressure fluctuation function is normalised with $\frac{U}{\sigma_p}$, however, no information is given concerning frequency scaling. This together with the lack of experimental data in the paper by Dumnov make it impossible reproduce the model or estimate how general the obtained spectrum actually is. However, it is encouraging to note that there appears to exist some universal features for pressure fluctuations in the separation region, for instance Gonzalez and Dolling showed that the spectra in the intermittent region, generated by different diameter cylinders, collapsing when plotting the magnitude $W_p(f)$ versus a reduced frequency given as $f \frac{L}{U}$. A comparison of intermittent power spectra for different basic flow cases was made in the present work, which showed that the low frequency pressure fluctuations are characterised by a Strouhal number of 0.07 based on the interaction length and the incoming free stream velocity. This indicates that the intermittent motion of the separation line is a generalised feature, common for various types of shock boundary layer interactions.

Other key elements in the Dumnov approach were investigated in the present work by applying the model on data obtained in tests with the LEA TIC nozzle by Girard and Alziary. Dumnov describes the intermittence by assuming a sinusoidal fluctuation between the two-pressure levels $p_i$ and $p_p$. The present analysis shows that this is an oversimplification, which overpredicts the average fluctuation level in the intermittent region. An accurate and physically more correct method is proposed here, based on the work by Kistler on external flow protuberances. In contrast to the Dumnov model, which only gives one constant value of the pressure fluctuation, the intermittence factor, which gives the fraction of time that the plateau pressure region is acting over the point of interest. According to Erengil and Dolling the error function shows a good fit to the intermittence factor, which means that the location of the separation shock has a Gaussian distribution within the intermittent region. This method was applied to the LEA TIC nozzle flow and shows a close agreement with test data.

On the whole, the basic idea of the Dumnov model constitutes a valid approach, which is supported by the present analysis. The correction and improvements suggested will contribute to the quantitative prediction of side-loads. The development and validation work of such improved models is currently ongoing at the different partners of the FSCD group, see e.g. the work by Volvo Aero in reference 108.

In general it can be said that appropriate parameters for normalising power spectra in the intermittent region still requires more work. In general reliable quantitative data on the structures and pressure fluctuations in the transverse direction are lacking and is fruitful area of future work.
12.2 NOZZLES WITH TRANSITION OF SEPARATION PATTERN

12.2.1 Observations in tests

It was observed already in the early 1970’s by Nave and Coffey⁹⁵ that a transition in separation pattern from the free-shock separation (FSS) to the restricted shock separation (RSS) and vice-versa might occur. However, it was not understood that these transitions also are the origin of two distinct side-load peaks, until Östlund⁶³ et al presented the detailed analysis of the VOLVO S1 nozzle flow. Östlund observed that when the flow reattaches and the separated region becomes enclosed by supersonic flow it is a sudden pressure drop of the plateau pressure and a subsequent jump of the separation point in the downstream direction. This transition is unsteady and asymmetrical, the reattachment of the flow occurs successively by occupying more and more of the nozzle wall, until the entire flow is reattached, i.e. the RSS condition prevails with a wall pressure distribution totally different from the one obtained at the previous FSS condition. When the reattachment point is close the nozzle exit, the flow begins to pulsate. The reason for this is the sudden increase in the plateau pressure behind the separation shock, which occurs when the former enclosed recirculating zone is opened up and ambient air is sucked in to the nozzle. This increase in pressure, forces the separation point to move upstream again, once more closing the recirculating zone. The procedure repeats itself until the increase of the feeding pressure is sufficient to move the reattachment zone out of the nozzle. This is known as the “end effect”. Östlund was able to correlate the occurrence of two distinct side-load peaks observed in test to the FSS-RSS transition and to the end effect respectively.

The observations and conclusions by Östlund⁶⁵ were the ignition for intensive research both within and outside Europe. Further sub scale experiments performed within different FSCD test campaigns⁶⁶-⁶⁸ as well as recent Japanese experiments⁷² confirm the transition mechanism for TOP and CTIC nozzles, both featuring an internal shock. In addition, full-scale engine tests for the Vulcain 1 engine have confirmed this mechanism as key driver during both start-up and shut-down⁷⁴.

12.2.2 Models for prediction

Stimulated by the test results, flow separation and side-load models for the transition from free- to restricted shock separation and vice versa have recently been developed within the European space community. A high accuracy have been achieved (6%) in matching model and experimental results. The key for this successful result is to predict the location where the transition takes place. The driving force for reattachment of the flow is when the radial momentum of the separated jet is directed towards the wall, which occurs only with a cap-shock pattern. By quantifying the momentum balance of the jet, the transition point can be determined.

On this basis, Östlund & Bigert⁶² simultaneously with Frey & Hagemann⁵³ proposed criteria for the prediction of transition from FSS to RSS. Both models account for the sudden pressure drop of the plateau pressure and the subsequent jump of the separation point when the flow reattaches and the separated region becomes enclosed by supersonic flow. Due to the complexity of the flow downstream of the reattachment point, which is characterised by subsequent compression and expansion waves, no models for this pressure recovery process exist so far. Instead a constant value of the plateau pressure based on test data experience is often used. This value is assumed to be constant until the RSS is transformed back into FSS and FSS criteria are applicable again. This transformation occurs either when the cap-shock is converted into the Mach disc or when the enclosed separation zone is opened up at the nozzle exit.

Based on these separation models ASTRIUM/DLR⁴ and VOLVO⁶¹ have developed side-load models by assuming that the initial transition from free- to restricted shock separation is the key side-load driver in TOP and CTIC rocket nozzles. The basic idea is that at the instant of transition a maximum side-load is expected if one half of the nozzle features FSS-, while the other half shows already RSS flow condition. For this case, the side-load calculation is squarely based on physical reasoning namely from a momentum balance across the complete nozzle surface. With this model the maximum aerodynamic side-load can be calculated.
12.3 AEROELASTIC EFFECTS

In highly aeroelastic cases a significant amplification of the side-loads can be obtained as the flow interacts with the mechanical structure. The study of aeroelastic effects in separated nozzle flows is rather complex, requiring dynamic models of the mechanical nozzle-engine support system, the flow separation, as well as the coupling between these two. A simple technique for handling these difficult coupling problems was proposed by Pekkari\textsuperscript{102,103} in the early 1990’s. The model consists of two main parts, the first dealing with the equation of motions of the thrust chamber as aerodynamic loads are applied, and a second part modelling the change of the aerodynamic loads due to the distortion of the wall contour. The wall pressure in the attached region is the nominal vacuum pressure profile with a pressure shift due to the displacement of the wall. In the original work by Pekkari, this pressure shift is determined with the use of linearised supersonic flow theory. However, experience has shown that this theory significantly overpredicts the pressure shift, and Östlund\textsuperscript{108} therefore proposed a modified approach where the pressure shift is extracted from 3D Euler simulations. The separation line in the nozzle is assessed with a simple separation criterion of Summerfield type (cf. Eq.28). The wall pressure in the separated region is assumed to be equal to the ambient pressure. This model predicts the aeroelastic stability, the modification of eigenfrequencies due to aeroelastic effects, as well as the transient behaviour during start up and shutdown of the nozzle. Different mechanical eigenmodes can be treated, however, from side-load point of view, the aeroelastic behaviour of the bending mode is the most relevant one. To verify this improved model a unique test set-up was designed by Östlund within the GSTP FSC test program.\textsuperscript{5} In order to resemble the bending mode of a real rocket engine nozzle, the model nozzle was flexible hinged at the nozzle throat, where the bending resistance could be changed with the use of exchangeable torsion springs. Thanks to the simple test set-up, the mechanical system could be described analytically (in contrast to real rocket engine cases, which must rely on complex FEM analysis) and the basic model assumptions could be verified separately. With these tests the model by Östlund was successfully verified. It was found that the method was able to reproduce the aeroelastic behaviour experienced in the VOLVO S1 and S6 nozzle tests and it was also shown that aeroelastic effects can be significant in week nozzle structures.

12.4 SCALING

Model experiments are necessary for the understanding of physical phenomena in nozzle flow as well as for validation of models for separation and side-load prediction. Such model experiments can only be successful if the designed sub-scale model is able to capture the most relevant physics of the full-scale rocket nozzle. The main challenge is to reproduce the behaviour of the chemical reacting hot propellants using air with totally different gas properties. Some basic ideas of the scaling considerations for the investigation of separation and side-loads in a real rocket nozzle has been presented in this thesis. The feasibility of two different scaling approaches has been examined by designing and testing two different sub-scale models, S1 and S3, of the Vulcain nozzle. In both cases, similarities of properties along the wall were considered. In addition to this, similarity in the overall flow field and shape of the internal shock was considered in the S3 nozzle. The chosen characteristic length was the nozzle exit radius, $r_e$, in sub-scale nozzle S1 and the nozzle throat radius, $r_t$, in sub-scale nozzle S3. The results and analysis of the scaling verify that the methodology used for scaling Vulcain to a sub-scale model operated with air has been successful. Both S1 and S3 have captured the relevant physical phenomenon found in Vulcain, especially the transition between FSS and RSS and vice versa and ensuing side-loads. The results from the model tests have been the basis for the understanding of the separation and side-load characteristics in TOP nozzles. They have, for the first time, revealed the transition between the different separation patterns as a basic mechanism behind for side-load generation. It must be kept in mind that the key for a good similarity between a sub-scale and full-scale nozzle is still the design of a representative sub-scale nozzle contour. The S3 nozzle shows a good similarity to Vulcain, since it reproduces the overall flow field and shape of the internal shock, in addition to flow properties along the wall. With this approach the side-load moment is accurately reproduced by combining the two length scales as $\frac{r_t^2}{r_e^2}$. The S1 nozzle, however, were only the wall properties were taken into account, is only capable of reproducing the basic separation and side-load phenomena of Vulcain.


R 26 Dumnov G. E., Nikulin G. Z., Ponomarev N. B., ”A study of Promising Rocket Nozzles”, (In Russian), NIITTP, YDK 533.697.4


R 28 Davis D. K., “Investigation of optimisation techniques for solid rocket motor nozzle contour”, AIAA 82-1188, 1982


R 30 Frandtl, L., ”'Uber Fluessigkeitsbewegung bei sehr kleiner Reibung” (On Fluid Movement at Low Friction, in German), International Mathematical Congress, Heidelberg, 1904

R 31 Prandtl, L., ”'Fuehrer durch die Stromungslehre’” (Text Book on Fluid Dynamics, in German), Friedrich Vieweg & Sohn, Braunschweig, 1960


R 60  Dolling D., Or C., “Unsteadiness of the shock wave structure in attached and separated compression ramp flows” Experiments in Fluids 3, Springer Verlag, 1985, pp. 24-32.
R 69  Robertson, J. E., “Prediction of In-Flight Fluctuating Pressure Environments Including Protuberance Induced Flows”, Wyle Laboratories Research Staff Rept. WR71-10, March 1971.


R 80 Darren, R., Hidalgo, H., “Fluctuating Pressure Analysis of a 2-D SSME Nozzle Air Flow Test”, In its Thirteenth Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion and Launch Vehicle Technology, Huntsville, Alabama, p 723-743 (SEE N96-29750 11-34), April 1995

R 81 Foster, C., and Cowles, F., ”Experimental Study of Gas Flow Separation in Overexpanded Exhaust Nozzles for Rocket Motors”, JPL Progress Report 4-103, May 1949


R 84 Campbell, C. and Farley, J., ”Performance of Several Conical Convergent-Divergent Rocket Type Exhaust Nozzles”, NASA TN D-467, September 1960


R 89 Lawrence, R.A., ”Symmetrical and Unsymmetrical Separation in Supersonic Nozzles”, Research report 67-1, Southern Methodist University, April 1967


R 91 Nasuti, F., and Onofri, M., ”Viscous and Inviscid Vortex Generation During Nozzle Flow Transients”, AIAA 96-0076, June 1996


R 94 Schmucker, R., ”Flow Processes in Overexpanding Nozzles of Chemical Rocket Engines” (published in German), Report TB-7,-10,-14, Technical University Munich, 1973


R 100 Carriere, P., "Remarques sur les Méthodes de Calcul des Effets de la Viscosité dans les Tuyères Propulsives" (Comments on computational methods of viscous effects in propulsion nozzles, in French), DGRR/WGLR Symposium, Bad Godesberg, also published as ONERA TP 408, October 1966


R 113 Lewis, J. E., Kubota, T., Lees, L., "Experimental investigation of supersonic laminar, two-dimensional boundary layer separation in a compression corner with and without wall cooling", AIAA J. 6, 7-14, 1968.


ACKNOWLEDGEMENTS

This work has been carried out at the Space Propulsion Division at Volvo Aero Corporation in Trollhättan within the GSTP program and the framework of the European Flow Separation Control Device (FSCD) working group. I would like to express my sincere thanks and appreciation to my supervisor Dr. Barbro Muhammad-Klingmann (Royal Institute of Technology), without whose guidance and support this work would not be possible. I would like to thank my close colleagues Dr. Lars Ljungkrona and Dr. Manuel Frey for their suggestions, remarks and inspiring discussions. Valuable help and suggestions from many people at Volvo Aero Corporation are appreciated and special thanks to those working at the Business Unit Nozzles. I would like to acknowledge all the FSCD members, especially Dr. G. Hagemann, Dr. M. Terhardt (ASTRIUM), Dr. M. Pons, Dr. P. Vuillermoz (CNES), R. Stark (DLR), Dr. R. Schwane, Dr. J. Muylaert (ESTEC), L. Torngren (FOI), Dr. Ph. Reijasse (ONERA), Dr. Ph. James (SNECMA) and Dr. Alziary de Roquefort (University of Poitiers) for the fruitful cooperation in this field. Last but not least I thank my wife Anna for her encouragement and understanding.

This work has been supported through grants from Volvo Aero Corporation and the Swedish Research Council for Engineering Sciences (TFR contract 285-98-717). These sources of support are gratefully acknowledged.