



Université Catholique de Louvain
École Polytechnique de Louvain



von Karman Institute for Fluid Dynamics
Turbomachinery and Propulsion Department

Thèse présentée en vue de l'obtention du grade de docteur en science de l'ingénieur

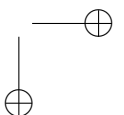
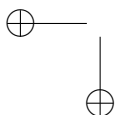
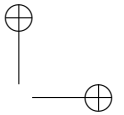
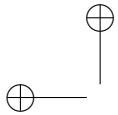
Experimental Study of the Performance and Stability of a Low Pressure Axial Compressor for a Contra-Rotating Turbofan Engine Architecture

Nicolas Van de Wyer

Décembre 2012

Composition du Jury

Prof. T. Arts (UCL, Promoteur)
Prof. J.-F. Brouckaert (VKI, Promoteur)
Prof. H. Jeanmart (UCL, Président)
Prof. P. Chatelain (UCL)
Dr. S. Hiernaux (Techspace Aero, Belgium)
Prof. X. Ottavy (Ecole Centrale de Lyon, France)



Abstract

The new contra-rotating turbofan architecture leads to a substantial reduction of the pollutants and the noise emission of civil aircrafts. This architecture is based on two shorter fans, rotating in opposite direction at lower speed. Due to the mechanical link between the fan and the low-pressure compressor, its speed is also reduced. In order to keep the pressure ratio constant, without weight increase, the low-pressure compressor stages are designed with an unusually high loading coefficient and flow coefficient, which leads to a reduced stability margin.

The purpose of this thesis is to present a complete characterization of the first stage of the low-pressure compressor for the new contra-rotating turbofan architecture in terms of global performance, unsteady flow field and instability phenomena. This work will be the basis for future research in this field, considering the low number of publications on low-pressure compressors and the absence of references concerning the low-pressure compressor of contra-rotating turbofans. The characterization is performed on a half scale model in the VKI R4 test facility. This facility is a closed loop, high speed compressor rig, allowing the investigation of the influence of the Reynolds number on the performance. The effect of a circumferential groove casing treatment on the performance, the stability and the unsteady flow field is studied. The measurements are compared to numerical simulations from various CFD codes using various turbulence models.

The very small size of the blades imposes a high level of miniaturization of the instrumentation, particularly challenging in the case of the unsteady measurements. The global performance of the compressor is derived from the steady conditions upstream and downstream of the stage at four operating points, from choke to near the stability limit. The simultaneous acquisition of the casing wall static pressure and the total pressure downstream the rotor allows a visualization of the tip leakage vortex in the three dimensions. The data processing developed for this visualization includes the phase lock averaging and a specific interpolation method taking into account the various phenomena driving the flow in these regions.

The performance of the compressor shows an important reduction of the pressure ratio between design and near stall conditions, caused by a flow separation along the suction side at the hub of the blade, typical of hub loaded compressor characteristics. At near stall conditions, the pressure defect region due to the flow separation covers 35% of the rotor pitch. The investigation of the instability in the compressor shows a modal stall inception process confirmed by several elements. The unstable regime is a rotating stall caused by the large separation at the hub of the blades. The stall cells' configuration in the compressor is completely unusual, with a single full-span cell splitting in multiple full-span cells, equally spaced and

iv

of equal size. In the single cell configuration, the stall cell covers 80% of the circumference. The non tip-critical aspect of the compressor makes the effects of the casing treatment on the mid-span performance and on the stall margin negligible. Nevertheless, the casing treatment induces a reduction of the blockage in the tip region when the origin of the tip leakage vortex is located upstream the casing treatment.

Acknowledgements

VITAL is a new collaborative research project, running for four years, which aims to significantly reduce aircraft engine noise and CO₂ emissions. It has a total budget of 90 million euros, including 50 million euros in funding from the European Commission. Snecma leads a consortium of 53 partners gathering all major European engine manufacturers: Rolls-Royce, MTU Aero Engines, Avio, Volvo Aero, Techspace Aero Rolls-Royce Deutschland and ITP, and the airframer Airbus. This work was carried out under SP3 Booster, Work Package 3.2. Advanced Aerodynamic Technologies. The author gratefully acknowledge the financial support of the European Commission.

This is for me the opportunity to thank all those who have made possible the achievement of this long-term job. First and foremost my dissertation directors: Professor Jean-François Brouckaert, who has been my mentor and my support for many years, and Professor Tony Arts, who gave me the opportunity to join the VKI family, for both my Master degree and my PhD.

My thanks also go to all the people who spent long hours with me down in the lab, cursing the R4, and above all trying to solve the numerous problems encountered. I would like to mention here Julien, Pierre and Jean-Christophe. I want to thank Terrence for the express probe-repairs, and all the workshop staff for their help.

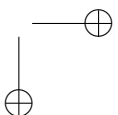
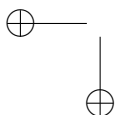
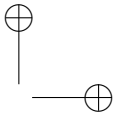
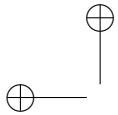
This dissertation includes a number of numeric results. These results were provided to me by Balazs Farkas of the VKI and by Jean-François Thomas of CENAERO. My thanks to both of them for having shared their work with me. My thanks also to Stéphane Hiernaux and Nicolas Isaac of Techspace Aero for the management of this project.

The VKI is a big family, and its spirit is exceptional. Thank you to all who have created this atmosphere and who allowed me to take the necessary breaks, essential for quality work. Julien, Delphine, Boris and Marcos. Thank you for the Japanese restaurants, the woks, the trattoria and other fast food counters; even if after these not too light meals my productivity was not at its best.

Thank you to all who, outside the VKI, helped me to forget about blades, performance, blockages and other turbo-pleasures for the time of one evening or a weekend. I paid the prize in lost neurones and headaches, but I sometimes really needed it.

A special thank to Baudouin who proofread this dissertation many times over.

And finally, thank you to you, Gaëlle and Choupinette, for this haven of peace and happiness that you were for me.



Contents

Nomenclature	ix
1. Introduction	1
1.1. European Aircraft-Engines Developments	1
1.2. The Challenge of Low-Pressure Compressors for Contra-rotating Turbo Fan Architecture	6
1.3. Objectives and structure of the thesis	8
2. Literature Survey	11
2.1. Secondary Flows in Compressors	11
2.1.1. Tip Leakage Vortex	12
2.1.2. Hub Corner Stall	19
2.2. Compressor Stability Problems	24
2.2.1. Rotating Stall	27
2.2.2. Surge	32
2.3. The Casing Treatments	33
2.3.1. History of Casing Treatments	33
2.3.2. Circumferential Groove Casing Treatment	37
3. Experimental and Numerical Tools	43
3.1. The VKI R4 Test Bench	43
3.2. The VITAL Test Section	46
3.3. Aero-Dynamic Design of the First Stage of the Booster for Contra- Rotating Turbofan Architecture	51
3.4. The Numerical Simulations Tools	52
3.5. Test Conditions	55
3.6. The Circumferential Groove Casing Treatment	56
4. Overall Performance of the Compressor Stage	59
4.1. Instrumentation for the Overall Performance Determination	60
4.1.1. Steady Pressure Measurements	60
4.1.2. Temperature Measurements	64
4.1.3. Flow Direction, Mach Number and Static Pressure Measure- ments	71
4.1.4. Three-hole Probe Measurements Uncertainty	73
4.1.5. Hot Wires	75
4.2. Inlet Stage Flow Conditions	77

4.3. Global Stage Performance for the Smooth Casing Configuration . .	86
4.4. Rotor Performance	95
4.5. Effects Of The Casing Treatment On The Overall Performance . .	100
4.6. Conclusion	103
5. Rotor Unsteady Flow Field	109
5.1. Unsteady Pressure Instrumentation	110
5.1.1. Casing Wall Pressure Sensors	110
5.1.2. Unsteady Total Pressure Probes	113
5.2. Casing Wall Static Pressure Contour Maps	114
5.2.1. Wall Fast Static Pressure Sensors Localization	114
5.2.2. Frequency Analysis	116
5.2.3. Post-processing procedure	117
5.2.4. Tip Leakage Vortex Path	125
5.3. Simultaneous Acquisitions	137
5.4. Conclusion	146
6. Stall Characterization	151
6.1. Specific Instrumentation	151
6.2. Stall Inception Process	152
6.3. Rotating Stall Patterns	156
6.3.1. Stall Cells Time Evolution	156
6.3.2. Stall Cells Characteristics	164
6.4. Conclusion	173
7. Conclusions and Perspectives	175
7.1. Conclusions	175
7.2. Perspectives	179
Appendices	181
A. Uncertainty Calculations	181
A.1. Thermocouple probe uncertainty	181
A.2. Three-hole probe uncertainty	181
B. Unsteady casing wall static pressure fields at choke conditions	185
Bibliography	189
List of Tables	199
List of Figures	201

Nomenclature

Acronyms

<i>CFD</i>	Computational fluid dynamics
<i>CR</i>	Relative to cruise conditions
<i>CT</i>	Casing treatment
<i>FFT</i>	Fast Fourier Transform
<i>IGV</i>	Inlet guide vane
<i>NN</i>	Nominal rotational speed
<i>PLA</i>	Phase locked average procedure
<i>PointA</i>	Choke conditions
<i>PointB</i>	Maximum pressure ratio at mid-span conditions
<i>PointC</i>	design conditions
<i>PointD</i>	Near-stall conditions
<i>PSD</i>	Power spectral density
<i>RANS</i>	Reynolds average Navier-Stokes
<i>TO</i>	Relative to take-off conditions
<i>VKI</i>	von Karman Institute for Fluid Dynamics
<i>WT</i>	Wavelet transformation
ACARE	Advisory Council for Aeronautical Research in Europe
BPR	By Pass-Ratio
CRTF	Low speed contra rotating fan for Contra-Rotating Turbo Fan
DDTF	Low speed fan for Direct Drive Turbo Fan
GTF	Low speed fan for Geared Turbo Fan
SRA	Strategic Research Agenda

x

Nomenclature

VHBR	Very High By Pass-Ratio
VITAL	enVironmenTALy friendly aero-engines

Greek Symbols

α	Absolute flow angle	[radian]
β	Relative flow angle	[radian]
δ_d	Displacement thickness	[m]
δ_m	Incompressible momentum-loss thickness	[m]
η	Efficiency	[%]
γ	Specific heat ratio	[-]
ω	Angular speed	[radian/s]
ϕ	Flow coefficient	[-]
$\phi_{covered}$	Percentage of circumference covered by a single stall cell	[%]
Ψ	Zweifel blade loading coefficient	[-]
ψ	Loading coefficient	[-]
ρ	Fluid density	[kg/m ³]
σ	Rotor solidity	[-]
θ	Camber angle	[radian]
φ	Phase shift	[radian]

Roman Symbols

$\dot{m}$	Mass flow of the engine exit jet	[kg/s]
Π	Total-to-total pressure ratio	[-]
A	Passage area	[m ²]
a	Sound velocity	[m/s]
AR	Blade aspect ratio	[-]
C_{ax}	Axial chord	[m]
D	Diffusion parameter	[-]
d	Diameter of the wire	[m]

Nomenclature		xi
D_f	Diffusion factor	[-]
D_m	Diffusion parameter from Yu and Liu [118]	[-]
DR	Diameter of the rotor	[m]
E	Voltage over a wire	[V]
f	Frequency	[Hz]
F_t	aircraft thrust	[kN]
H	Enthalpy	[kJ/kg]
h	Heat transfer coefficient	[W/m ² .K]
i	Incidence angle	[radian]
I_w	Current going through the wire	[A]
k	Thermal conductivity coefficient	[W/m.k]
l	Length of the wire	[m]
M	Mach number	[-]
MEI	Maximum efficiency improvement	[%]
N	Mechanical rotational speed of the compressor	[RPM]
n	Number of elements	[-]
Nu	Nusselt number	[-]
O	Offset of the calibration curve	[unit]
P	Pressure	[Pa]
P_s	Static pressure	[Pa]
P_t	Total pressure	[Pa]
Pr	Prandtl number	[-]
q	Dynamic pressure	[Pa]
Q_{in}	Heat dissipated in a Hot-wire	[W]
R	Perfect gas constant	[J/K.Mol]
r	Recovery factor	[-]
R_w	Resistance of the wire	[Ω]

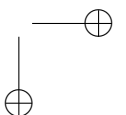
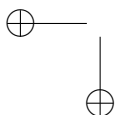
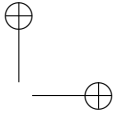
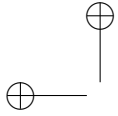
xii Nomenclature

Re	Reynolds number	[-]
S	Hub corner indicator	[-]
S_w	Stall indicator	[-]
SF	Incompressible shape factor of the boundary layer	[-]
Sl	Slope of the calibration curve	[unit/V]
SM	Satll margin	[%]
SMI	Stall margin improvement	[%]
T	Temperature	[K]
t	Time	[s]
T_0	Flow total temperature	[K]
T_f	Flow temperature	[K]
T_j	Thermocouple junction temperature	[K]
T_s	Flow static temperature	[K]
Tu	Turbulence level	[-]
U	Peripheral speed	[m/s]
V	Voltage	[V]
V_0	Velocity of the aircraft	[m/s]
V_j	Velocity of the jet	[m/s]
w	Relative velocity	[m/s]
W_N	Leakage velocity	[m/s]
X_n	Complex Fourier coefficients	[-]

Subscripts

01	Total conditions at the inlet
1	Relative to the inlet
2	Relative to the outlet
acq	Relative to the acquisition
atm	Relative to the atmospheric conditions

<i>c</i>	Relative to the central element
<i>calib</i>	Relative to the calibration
<i>CR</i>	Relative to the rotation of the stall cell
<i>free</i>	Relative to the free stream
<i>hub</i>	Relative to the hub
<i>L</i>	Relative to the left element
<i>mean</i>	Mean value
<i>mid</i>	Relative to the mid-span
<i>OSC</i>	Relative to the pressure variation
<i>probe</i>	Measured value
<i>prop</i>	relative to the propulsion
<i>ps</i>	Relative to the pressure side
<i>R</i>	Relative to the right element
<i>red</i>	Reduced value
<i>reference</i>	Reference value
<i>RMS</i>	Root-mean-square value
<i>smooth</i>	Relative to the smooth casing configuration
<i>SoA2000</i>	State-of-the-art in the years 2000
<i>ss</i>	Relative to the suction side
<i>tank</i>	Relative to the settling chamber conditions
<i>treatment</i>	Relative to the configuration with casing treatment
<i>up</i>	Relative to the upstream conditions



Chapter 1.

Introduction

1.1. European Aircraft-Engines Developments

The prediction of the evolution of the global air traffic announces an annual growth of around 5% corresponding to an annual increase of 3% of the CO_2 emissions. This leads to an important increase of the impact of air traffic on the environment as well as on human society. Nowadays, the major challenge of the European aviation industry is to cover the growing demand of economical and safe air travel and, at the same time, to reduce the global impact of air traffic on the environment. These objectives will be reached via a permanent improvement of the aircraft technologies based on a long-term view.

In the Strategic Research Agenda (SRA) published in 2002, the Advisory Council for Aeronautical Research in Europe (ACARE) describes the requirement of research for the aeronautic industry for 2020. The SRA identified five major challenges to be investigated: Quality and Affordability, Environment, Safety, Air transport System Efficiency and Security. Concerning the environment, ACARE fixed, amongst others, the following objectives for the overall air transport system, including the engine, the aircraft and operations:

- Reducing fuel consumption and CO_2 emissions by 50%, with 20% for the engine alone,
- Reducing perceived external noise by 50%, with 6 dB per operation for the engine alone by an improvement of the engine acoustics,
- Reducing NOx by 80%, with 60 to 80% for the engine alone by improving the burner technology. [107]

During the last thirty years, the common tendency of the commercial aircraft turbofan design improvements has been to increase the bypass ratio (BPR), leading to a reduction of the fuel consumption by improving the propulsive efficiency η_{prop} . The propulsive efficiency is given by [100]:

$$\eta_{prop} = \frac{2}{1 + \frac{V_j}{V_0}} \quad (1.1)$$

where V_j and V_0 are respectively the velocity of the engine exhaust jet and of the aircraft. To achieve a 100% efficiency the two velocities should be equal. The aircraft thrust F_t (based on Newton’s 2nd law) is given by:

$$F_t = \dot{m} (V_j - V_0) \quad (1.2)$$

with $\dot{m}$ is the mass flow of the engine exit jet. An engine producing thrust requires a velocity ratio $\frac{V_j}{V_0}$ superior but close to unity. As a consequence, a higher By-Pass ratio for a constant thrust engine allows a reduction of the velocity difference due to the increase of $\dot{m}$, and thus an improvement of the propulsive efficiency. Furthermore the increase of the BPR has a strong reduction effect on the two major contributors of engine noise: the fan noise and jet noise. As explained above, the use of a higher BPR allows a reduction of the jet velocity, which is the main factor of jet noise. This reduction is attained by decreasing the pressure ratio of the fan, which is achieved by a reduction of its rotational speed, main factor of fan noise. Currently the increase in BPR has reached its limit in terms of reduction of fuel consumption [2]. Although a higher BPR offers a clear reduction in specific fuel consumption, it also leads to a significant increase of engine weight as well as of nacelle and installation drags. The optimum BPR must be found between the benefit in terms of fuel consumption provided by a higher BPR and the penalties in terms of weight and drag. Based on available technologies this optimum is currently around 7 to 9. Because of this limitation the objectives fixed by ACARE 2020 cannot be reached by the on-going evolution of technologies, and therefore requires a decisive breakthrough in technology. This breakthrough should allow developing a new set of technologies for producing very high By-Pass ratio (VHBR) engines, reducing noise emissions and fuel consumption of the engines with minimal engine drag and weight penalties. This is proposed in the VITAL project.

The FP6 European research project VITAL (enVIronmenTALy friendly aero-engines) is aiming at a breakthrough in low noise and low emission engine architectures. The main objectives of VITAL are to develop and validate radical and innovative engine structures and architectures that alone will provide a:

- 6 dB noise reduction per aircraft operation,
- 7% reduction of CO_2 emissions.

compared to engines in service prior to 2000 such as the CFM56/7 and the Trent 772B.

These objectives are achieved by developing and validating novel lightweight and low noise technologies for commercial aircraft engines and thus providing a key step towards achieving ACARE environmental goals in terms of CO_2 and noise. As the weight increase is driven by the evolution of the low-pressure system components, due to the changes in the engine diameter, VITAL focuses on these components

1.1. European Aircraft-Engines Developments

3

with the objective of reducing the weight of every low-pressure component by 25 to 30%. The main objectives of VITAL will be achieved by the development of the following innovative technologies: new high- and low-speed low-pressure compressor concepts and technologies, lightweight structures using new materials, low-pressure turbine technologies for weight- and noise reduction, advanced low-pressure torque shaft and overall engine assembly. These technologies are therefore applied to three different innovative fan architectures:

- Low speed fan for Direct Drive Turbo Fan (DDTF)
- Low speed fan for Geared Turbo Fan (GTF)
- Low speed contra rotating fan for Contra-Rotating Turbo Fan (CRTF)

Fig.1.1 presents these three architectures designed, manufactured and rig scale tested in the VITAL project as well as the various new technologies developed and investigated. Some of these technologies are generic and usable in the three engine types, while others are specific to one or other of the architectures.

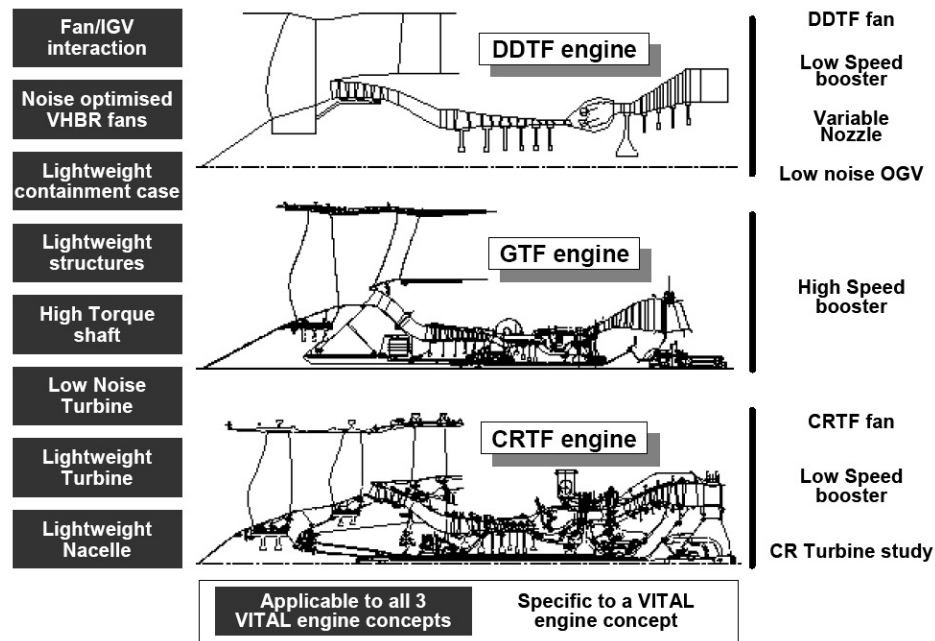


Figure 1.1.: Innovative VITAL engine architectures and developments (generic and specific to the type of engine) [2].

The **DDTF** is typically the current double shaft and double flow engine architecture. The improvement of the competitiveness of this architecture is mainly

based on a very high By-Pass ratio low-speed fan with high aerodynamic loading, coupled with variable nozzle technology, including intensive use of lightweight material.

The **GTF** is based on the use of a speed reduction gear on the low-speed shaft of a two-shaft engine, with the fan on the low-speed side, and the low-pressure compressor and turbine on the high-speed side [113]. Decoupling the fan rotational speed from the booster and low-pressure turbine speed gives an additional degree of freedom in the optimization of the aircraft engines. Indeed, as presented above, the fan speed in a VHBP is reduced, entailing a speed reduction of the low-pressure compressor and turbine. The conversion of shaft power into potential energy for the compressor, and inversely for the turbine, at lower rotational speed requires a larger number of stages. Additionally, the torque transmission requirement for the low-pressure shaft increases. This induces an increase of length, weight and cost of the low pressure part of the engine. By means of a Fan Drive reduction Gear System the fan speed is decoupled from the rest of the low-speed shaft, allowing the low-pressure compressor and turbine to run at higher and more appropriate speeds. The additional weight of the gear system is more than offset by the reduction of the number of stages of the low-pressure turbine compared to a direct drive turbofan.

In a **CRTF** two fans are rotating in opposite directions instead of one, as in the conventional aircraft engine architecture. These fans are driven by a contra-rotating low-pressure turbine, entailing a three-shaft architecture. The low-pressure compressor is coupled with the downstream fan. This configuration allows a reduction of the rotational speed of the fans as well as a reduction of the tip fan radius, inducing a decrease of the fan tip speed, which is the main contributor of the fan noise. As in VHBP engine the fan noise is the predominant noise source since there is a reduction of the exhaust jet noise. This fan configuration leads to an important reduction of engine noise by a decrease of the fan noise. The reduction of the fan rotational speed for the same aerodynamic loading is roughly 30% compared to a conventional engine architecture. In terms of emissions this configuration introduces a reduction of the engine size, owing to the moderate tip fan radius, implying a reduction of the engine weight and drag. This leads to a decrease of the engine consumption and thus of emissions. This new set of technologies and innovative architectures will enable the aero-engine industry to achieve the ACARE 2020 objectives by the use of VHBR engines ($BPR = 9 - 12$) operating with a significant reduction in both noise and fuel consumption. Figure 1.2 shows the evolution of the noise emitted by the aircraft engine in function of the By-Pass ratio. The trend is clearly to increase the BPR in order to reduce the engine noise. The *CRTF* architecture presents a similar reduction of the noise for a lower BPR thanks to the reduction of the fan tip radius. Figure 1.3 shows the fuel consumption in function of the BPR. An optimal BPR giving the minimum fuel consumption is observed. This optimum corresponds to the best balance between the propulsive efficiency improvement by the increase of the BPR and the drawback of the engine weight and drag increase due to the increase of the fan tip

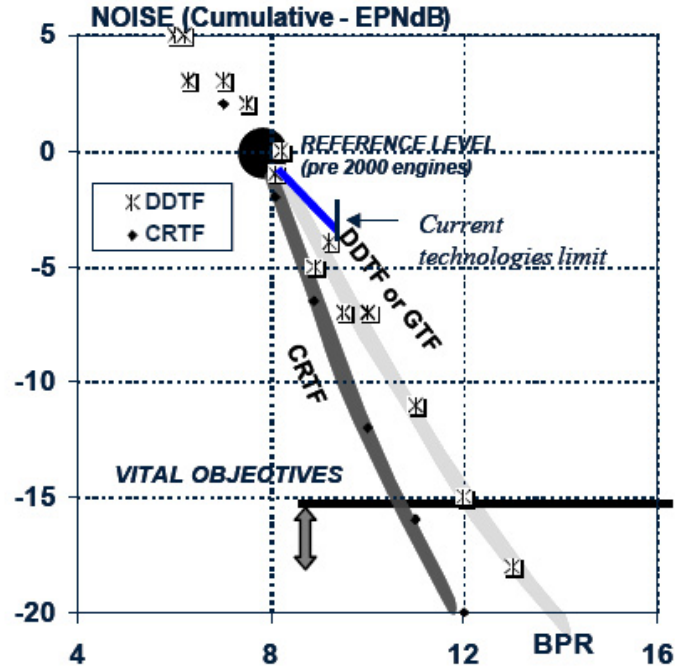


Figure 1.2.: Noise evolution in function of the By-Pass ratio.

radius. This optimal BPR is actually around 9 but the development of the various technologies considered in the VITAL project will allow raising this optimum to around 12.

Currently, another architecture of aircraft engine presenting a high potential for fuel consumption reduction is investigated. It is the open-rotor. In this architecture, the fan is not enclosed within the nacelle, leading to a high reduction of the size of the latter. This allows an important decrease of the weight of the engine and of its drag, meaning a 25 – 30% reduction of the engine’s fuel. This architecture can be improved by the use of a counter-rotating fan, allowing an increase of the engine’s effective thrust, via a reduction of the swirl of the air leaving the fan, and a reduction of the noise via a reduction of the engine’s rotational speed. Nevertheless, several challenges are associated with the development of the open-rotor architecture. These challenges are: the noise, the safety of the passengers, a limited fly velocity, the very diameter of the fans, and the accessibility and maintenance issues.

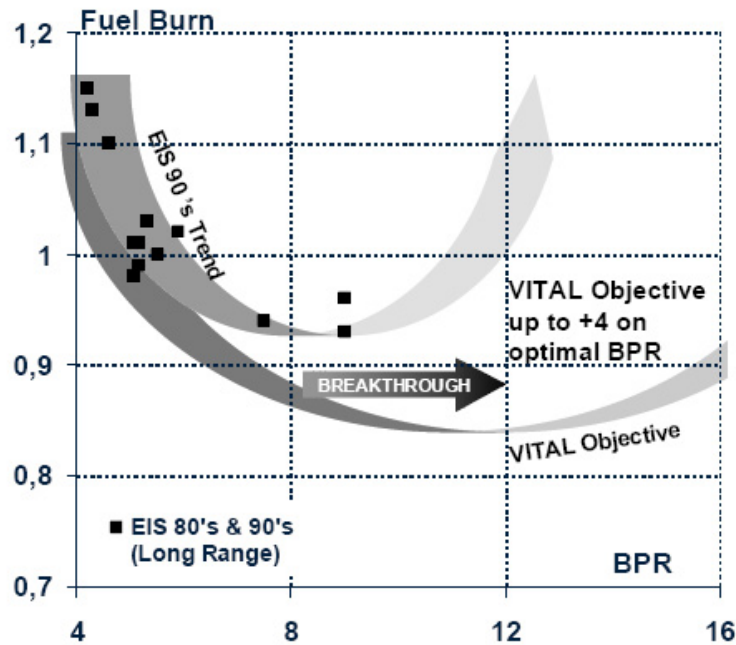


Figure 1.3.: Fuel consumption evolution in function of the By-Pass ratio.

1.2. The Challenge of Low-Pressure Compressors for Contra-rotating Turbo Fan Architecture

The innovative engine architecture CRTF is based on the use of two contra-rotating fans with a lower rotational speed compared to a classical engine. This characteristic has a strong influence on the low-pressure compressor, or booster, located downstream the fans. The first consequence of this architecture on the booster comes from its mechanical coupling with the downstream fan. The reduction of the rotational speed of the fan induces a lower rotational speed for the booster. This implies a decrease of the pressure ratio reached by the booster under equivalent conditions. In order to keep the pressure ratio constant, two solutions exist. The first one is to increase the number of stages of the booster. This solution induces an increase of both the length and the weight of the booster, leading to an important increase of the weight of the engine. This leads to an increase of the fuel consumption, a result contrary to the objectives of VITAL. Furthermore it also implies an increase of the production cost of the engine. The second solution is to keep the same number of stages for the booster, with a higher loading. This solution allows the booster to perform an equivalent pressure ratio as in a

1.2. The Challenge of Low-Pressure Compressors for Contra-rotating Turbo Fan Architecture 7

classical engine at lower rotational speed, without any cost or fuel consumption penalty. With this solution and for the same pressure ratio, a CRTF booster presents a much higher loading coefficient, $\psi = \Delta H/U^2$, as well as a higher flow coefficient, $\phi = V_z/U$ [51]. Figure 1.4 shows the evolution of load- and flow coefficient in comparison with a typical 4-stage booster design as state-of-the-art in the years 2000 (SoA2000). An advanced 3-stage high-loaded booster with identical

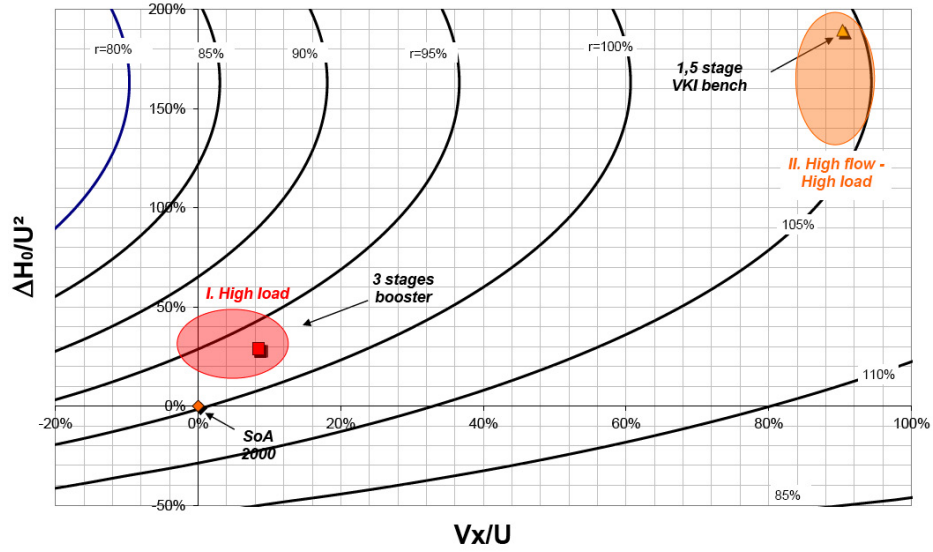


Figure 1.4.: Phi/Psi diagram of booster stages.

pressure ratio as the CRTF booster are presented. The positioning of the CRTF booster design is in an unusual region of very high load and very high flow coefficients. The parametric curves show a slight reduction of the deceleration ($r = (w_2/w_1)/(w_2/w_1)_{SoA2000}$) for the CRTF booster. This induces a lower staggering of the blades and relatively higher axial chords.

The second consequence is the reduction of the aspect ratio. Being located after two fans instead of one, the mean radius of the CRTF booster is higher. This allows an increase of the circumferential speed of the blades and thus a slight reduction of the load factor. The higher mean radius implies a higher hub/tip ratio for the same mass flow. CRTF booster blades have therefore a lower aspect ratio.

The high loading of the booster for the CRTF architecture induces a limitation of the stability range of the compressor as well as a decrease of its efficiency. The high loading leads to a higher risk of flow separation, and thus a reduction of the stability margin of the compressor, and to an increase of the secondary flows in the stage [90], associated with a reduction of the efficiency. This efficiency reduction

is further aggravated by the high sensitiveness to the secondary flows of stages characterized by low aspect ratio.

One of the challenges of the development of the innovative counter-rotating turbofan architecture is thus the design of the low-pressure compressor. The booster has to reach, at lower rotational speed and lower aspect ratio, the same pressure ratio with acceptable efficiency and stability margin.

1.3. Objectives and structure of the thesis

The present work is a large investigation of the characteristics of the first stage of a booster, typical of the new contra-rotating turbofan architecture. Today, the number of publications about the stability, performance and aerodynamics of the low-pressure compressor for conventional turbofans is really limited. And there are no publications on the low-pressure compressor for a CRTF with high ψ and ϕ . This thesis will thus compensate this lack of references on the subject.

The first part of this work was the installation of the experimental setup. This includes the refurbishing and the modernization of the VKI-R4 facility that was unutilized for 20 years. It comprises also the development of all the specific instrumentation and the post-processing procedures for the data reduction.

The global performance and the stability limit of this particular design, characterized by a high load coefficient, a high flow coefficient and a low aspect ratio, are determined experimentally.

A detailed experimental survey of the unsteady phenomena occurring in the blade passage of the rotor row has been performed. The importance of these phenomena comes from their weight in the loss generation in the compressor and thus their big influence on the global performance of the stage. Furthermore, these phenomena may be at the origin of the instability of the compressor. A particular attention is given to the tip leakage vortex.

The investigation of the influence of the use of a circumferential groove casing treatment on the global performance and on the tip flow field is presented. This investigation is based on the comparison between the experimental data acquired with the smooth casing and the casing treatment configurations. This allows a better understanding of the effect of casing treatment on the flow field in order to help the design of the future casing treatment.

The experimental data are compared with the results of numerical simulations performed with two CFD codes with various turbulence models. This allows to assess the numerical capability to predict the global performance, the stability limit, the unsteady phenomena and the effect of an axisymmetric casing treatment in a low-pressure compressor.

Chapter 2 gives a review of literature concerning the various concepts and physical phenomena addressed in this thesis. The works previously published about secondary flows in compressors, stability problems of the compressors and casing treatments are summarized.

1.3. Objectives and structure of the thesis

9

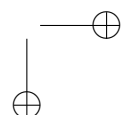
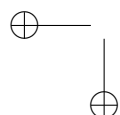
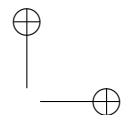
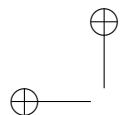
The various experimental test campaigns, which are the core of this work, were performed in the VKI-R4 test facility. The latter is described in chapter 3. Furthermore, this chapter presents the details about the test section, the conditions of the various tests as well as the parameters of the numerical simulations used for the comparisons.

The establishment of the overall performance of the compressor is presented in chapter 4. Firstly, the instrumentation used is described. The global performance of the compressor as well as the stability limit under cruise and take-off conditions are detailed. And finally the effect of the casing treatment on the performance and the stability of the compressor is shown. The experimental results are presented simultaneously with the numerical simulations.

Chapter 5 presents the results of the investigation of the unsteady flow field around the rotor. The tip leakage vortex path is characterized by using a combined visualization of the unsteady casing-wall static-pressure and the total pressure downstream the rotor. The effect of the circumferential groove casing treatment on the unsteady flow field is detailed. In this chapter, the importance of the hub corner flow on the blockage is evidenced.

In chapter 6, a complete characterization of the stall phenomena occurring in the compressor is presented. After a description of the specific instrumentation used, the stall inception mode is described. Then, the geometrical characteristics of the stall cells as well as their evolution in time is detailed.

This thesis is completed with chapter 7, which comprises the main conclusions of this work.



Chapter 2.

Literature Survey

2.1. Secondary Flows in Compressors

As set out in the previous section, the booster blades of a CRTF architecture are characterized by a lower aspect ratio and a high loading. As mentioned above, these two parameters have a strong influence on the stability and the overall performance of the stage through an increase of the importance of the secondary flows.

In a turbomachines, secondary flows are generated by the presence of the end-wall boundary layers and the turning of the flow in the blade passage [97]. Secondary flows are relatively minor flows, superimposed to the primary flow. The two main factors inducing the secondary flows in a turbomachine are:

- the presence of a clearance between the blade and the casing, in the case of an un-shrouded rotor or stator, allowing a flow leakage from the pressure side to the suction side of the blade, inducing the tip leakage vortex.
- the viscous boundary layer of the fluid along the endwalls, causing an important modification of the flow direction in the boundary layer. This leads to the appearance of various secondary flows and vortices. One consequence of these secondary flows is the hub corner separation or hub corner stall.

The tip leakage flow and the related vortex as well as the hub corner separation are the principal phenomena causing the deterioration of the overall performance of a compressor. The blockage they introduce is responsible for a limitation of the maximum pressure-rise and mass flow reached by the compressor. The mixing related to the presence of these phenomena leads to the generation of entropy, inducing a decrease of the efficiency. Furthermore, the tip leakage- and hub corner flows may drive the compressor to an unstable regime like rotating stall and surge. Since instability can lead to a dramatic deterioration of the pressure rise, and in some instances to the destruction of the compressor, a stability margin is a prerequisite. In case the initial margin would be too small, it can be extended by using various technologies like the casing treatment.

2.1.1. Tip Leakage Vortex

Turbomachines are composed by a series of fixed (stators) and mobile parts (rotors). Mechanical contacts between these two parts in relative motion are to be avoided at any time. Indeed, such contacts may lead to substantial damage of the blades, introducing an important deterioration of the machine performance and, in the worst case, its total destruction. A clearance between the tip of the rotor blades and the casing wall is hence always present in a turbomachine. This clearance is called the tip gap. The height of this gap has to be high enough to guarantee a positive gap at every operation of the engine. Indeed in an aircraft, the engine is submitted to important transient regimes at start-up, strong accelerations or landing. During these operations, the thermal and dynamic responses of the different parts of the engine (casing, rotor blades, disk,...) are different. This induces a variation of the tip gap between 0.5 and 3.5% of the chord [37].

On the other hand, the presence of the tip clearance has a positive impact on the flow in the tip region. A tip gap and the related leakage flow allow a decrease of the separation occurring at the tip corner of the blade (Cumpsty [22]). The phenomenon of corner separation will be presented in section 2.1.2. The positive effect is increasing with the size of the gap.

Nevertheless, the tip clearance introduces blockage and mixing processes that lead to a deterioration of the performance and stability of the compressor, and the negative effect is depending on the clearance height. This is illustrated on Fig.2.1 from Tan [106]. The performance map of a high-speed six-stages compressor is presented for a tip gap varying between 1 and 3.4% for all rotors. At nominal speed, an increase of the tip gap introduces a dramatical drop of pressure-ratio of 10% for a tip gap height variation of 1 to 2%, and of 25% for a variation of 1 to 3.4%. In terms of efficiency, the variation of the tip gap from 0.3 to 0.7mm seems not to influence the compressor. However, for a tip gap of 1.1mm the efficiency is 2.5% lower. Furthermore the stability margin is considerably reduced by increasing the tip gap. The stability limit corresponds to the stable point at the lowest mass flow before an unstable regime, as it will be presented in section 2.2. Since an unstable regime has to be absolutely avoided, in order to preserve the engine integrity, a security margin, called stall margin, is taken between the operating point and the stability limit. On Fig.2.1 the stability limit is moved towards higher mass flows, inducing a significant decrease of the stall margin. A large tip gap leads clearly to a considerable deterioration of the aerodynamic performances of the compressor.

The determination of the nominal tip gap is then a compromise between the mechanical constraints for the preservation of the engine integrity and the conservation of a high aerodynamic performance of the compressor. From an aerodynamic point of view there is an optimum size, balancing the favorable impact of the leakage flow on the corner separation and the loss induced by this flow. For small size blades this optimum is generally lower than the mechanical constraints of the compressor, and the effect on the corner separation is then generally not considered. The nominal tip gap height is about 1% of the blade chord in modern

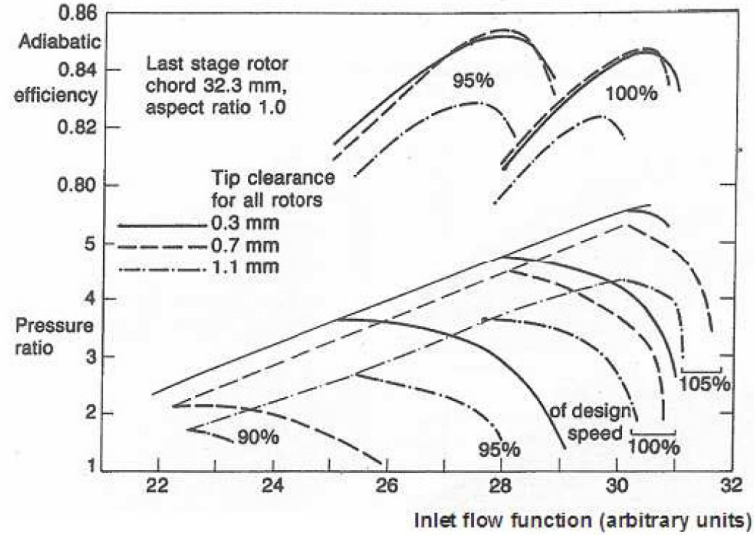


Figure 2.1.: Tip gap effects on the overall performance and stability of a high-speed six-stage compressor [106].

low-pressure axial compressors. A last point concerning the tip gap height is its evolution with the engine operating time. The wear and tear of the engine during its life results in an increase of the tip gap with the number of cycles of the engine. This leads to a deterioration of the overall performance of the engine. A regular overhaul of the engine is thus required.

The influence of the tip gap on the overall performance of a rotor is due to the scraping vortex and to the tip leakage flow. The scraping vortex is caused by the relative motion between the blade and the casing wall boundary layer. This induces a blockage of the annulus casing wall boundary layer, located outside of the tip gap by the pressure side of the blades and its deviation into the scraping vortex. The tip leakage flow is a kind of jet, developing through the tip gap, driven by the pressure difference between the pressure side and the suction side in the tip region. At the interface between the leakage flow and the main flow at the outlet of the clearance, the two flows have a different velocity direction with similar intensity, inducing the presence of a thin vortex sheet between them. This vortex sheet rolls up to create the tip leakage vortex. The onset of the vortex occurs at the location of the highest loading, meaning the highest vorticity of the sheet [58]. Figure 2.2 shows the tip leakage flow and its resulting vortex. Among the two vortices presented above, the tip leakage vortex is the predominant factor of the loss generated in a rotor.

The negative impact of the tip leakage flow and the vortex on the global per-

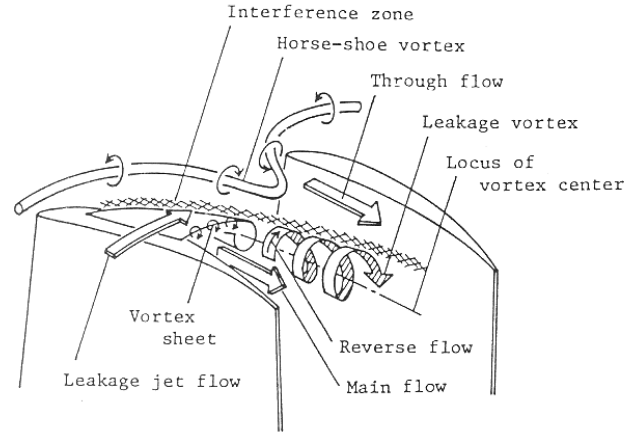


Figure 2.2.: Tip leakage flow and resulting vortex from [58].

formance of a compressor as well as the parameters influencing the importance of this impact have been intensively studied in the last 55 years. In 1954, Rains [89] has presented a theoretical and experimental study of the tip clearance flow in an axial compressor. In his thesis, Rains already pointed out the losses induced by this flow and the resulting vortex system. According to Lakshminarayana [67] the tip region contributes for a third to a half of the losses in turbomachinery. The substantial character of the contribution of the tip clearance flow in the losses brings some comprehension of its interaction with the main flow, as well as some knowledge of the parameters influencing its contribution. The loss of performance in the tip region due to the tip leakage flow are caused by three phenomena: a reduction of the work in the tip region, entropy generation in the mixing region between tip leakage and mainstream flows, and a blockage in the blade passage induced by the tip leakage vortex.

The decrease of work at the blade tip occurs because the blade loading drops off towards the tip [30] due to the pressure-rise reduction by leakage from the pressure side to suction side. This pressure-rise reduction is explained by the leakage flow velocity. The leakage velocity is defined as the component of the relative flow velocity normal to the blade surface at the exit of the tip gap, W_N . The amplitude of W_N is directly depending on the pressure difference between pressure- and suction sides. Thus, W_N is low at the trailing- and leading edges and maximum between 25 and 50% of the axial chord. On Fig.2.3 Lakshminarayana [66] shows that the presence of the leakage velocity leads to a deviation of the flow away from the blade. The turning in the tip region is consequently lower than in the main flow as well as the work performed in this region. This work shortage leads to a decrease of the global pressure ratio carried out by the blade row.

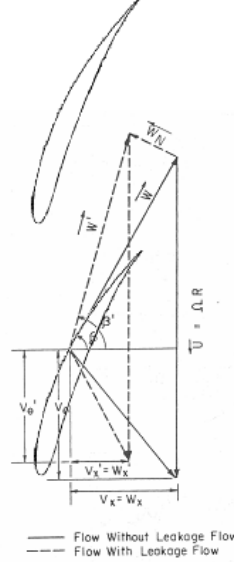


Figure 2.3.: Velocity triangle in the tip clearance [66].

A second source of performance loss due to the tip leakage vortex is the entropy production by the mixing phenomenon. This phenomenon is occurring at the interface between the tip leakage flow jet and the main flow on the suction side of the tip gap. These two flows have more or less the same velocity magnitudes but totally different directions. This induces a mixing process leading to an increase of the entropy and hence a loss. According to Denton [29] the total entropy produced in this mixing process is a function of the tip leakage flow rate and the flow velocity difference between the two blade surfaces. The high interaction between the tip leakage jet and the main flow leads to a very high turbulence level at the flows' interface.

The tip leakage flow and the related vortex introduce an important blockage phenomenon in the tip region. The blockage is defined as a reduction of the effective flow passage area, induced by a velocity deficiency. Khalid et al. [61] presented the blockage as similar to the displacement thickness related with the boundary layer. In the case of blockage, the velocity component considered for the calculation is the one in the direction of the main flow and the integration is performed over the defect region. This region is defined based on the velocity gradient.

On Fig.2.1 the influence of the tip gap size on the overall performance of the compressor is clearly visible. In order to understand this influence it is funda-

mental to know its effect on the flow field. In 1989, Inoue and Kuromaru [58] investigated the wall static pressure distribution above a single rotor as well as its fluctuation. The measurements were performed with a movable high-response pressure transducer, and a phase-locked method was applied to the pressure signals. The authors showed the presence of a low-pressure zone along the blade suction side as well as a trough of wall pressure starting at the low-pressure zone and extending downstream in the flow passage. The low-pressure zone corresponds to the onset of the rolling-up of the tip leakage vortex, and the vortex axis is aligned with the center of the trough in the wall pressure. Inoue and Kuromaru observed that an increasing tip gap leads to the position of the low-pressure zone moving downstream. These observations were confirmed by the work of Storer and Cumpsty [102] in 1991. They compared the experimental and numerical endwall static pressure distribution of a linear cascade, Fig.2.4. On this figure the movement

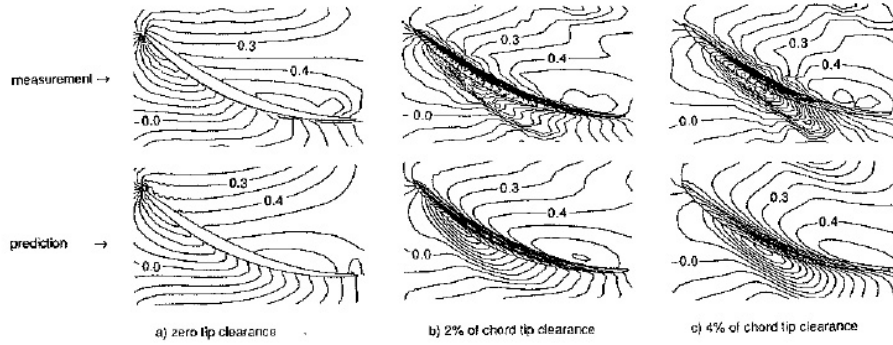


Figure 2.4.: Casing wall pressure coefficient of a rotor in a multistage axial compressor for different tip gap size [102].

downstream of the leading edge of the low-pressure zone in function of the tip gap size is clearly visible. Using a flow visualization by means of oil they observed the coincidence of the tip leakage vortex with the pressure trough at the casing. Using a comparison between experimental and computed data Chen et al. [17] showed the independence of the tip leakage vortex direction from the tip gap. The effect of the tip gap clearance on the overall performance was presented by Khalid et al. [61]. They determined numerically the blockage in the tip region, in the hub region and over the full span in a low-speed compressor for clearances equal to 1.4 and 3% of the chord. The blockage in function of the mass flow is presented in Fig.2.5. The growth of the total blockage due to a clearance increase is clearly related to the tip blockage rise.

The amplitude of the velocity of the leakage flow is directly linked to the pressure difference between the two sides of the blade in the tip region, *i.e.* the tip blade loading. The leakage velocity follows the pressure distribution along the blade.

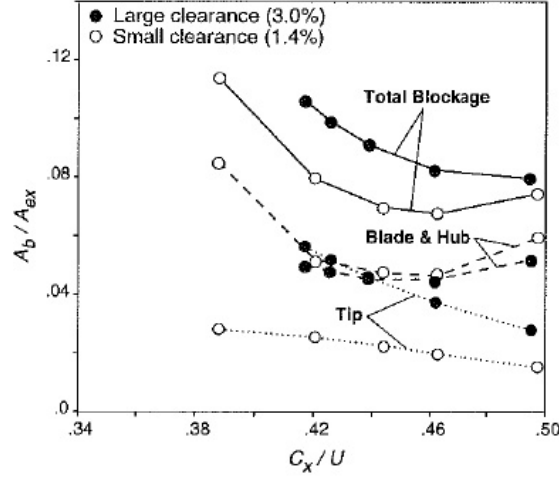


Figure 2.5.: Influence of the clearance on the blockages in a low speed compressor [61].

Typically, at design point, the leakage velocity has a small amplitude near the leading edge, reaches a maximum value between 25% and 50% of the chord and then decreases to the trailing edge. This velocity distribution is shown by Storer and Cumpsty [102]. The tip loading is depending on the design of the blade as well as on the operating flow coefficient. The impact of the flow coefficient on the vortex path is very important. Chen et al. [17] presented a comparison between experimental and numerical tip leakage vortex paths for various flow coefficients from design to stall point. They observed that an increase of the loading leads to a movement of the vortex away from the blade suction surface. Lu et al. [78] explained this behaviour by the fact that the interface between main flow and tip leakage flow results from a balance between the momentum of these two flows. The reduction of the flow coefficient induces a reduction of the main mass flow and an increase of the tip leakage due to the higher loading. The result is a movement of the tip leakage vortex towards the leading edge plane. In 2006 Bergner et al. [9] investigated the tip vortex path of a transonic axial compressor. They established the static pressure field by using high frequency response pressure transducers. The ensemble average wall pressure at design and near stall are presented on Fig.2.6. The tip leakage vortex path is indicated by the dashed line. The origin of the tip leakage vortex corresponding to the low-pressure zone moves upstream with increasing loading. At design operating point (on the right hand side) the origin of the tip leakage vortex corresponding to the low-pressure zone is located at around 15% of the chord. Near stall (on the left hand side), the origin is located

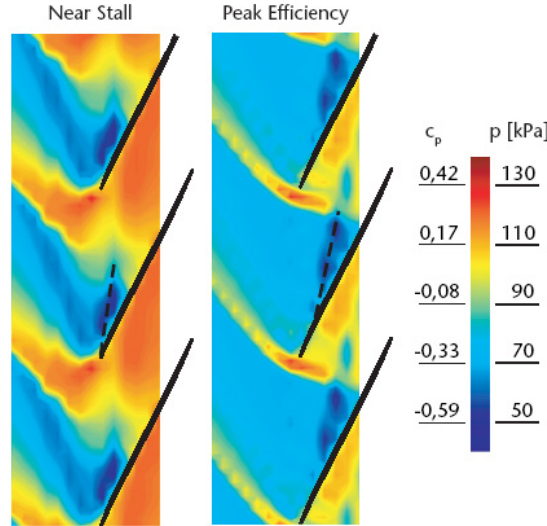


Figure 2.6.: Wall static pressure field of a transonic axial compressor under design and near stall conditions [9].

at the leading edge of the blade. The increase of the blade loading leads to an upstream displacement of the vortex origin. The increase of the inclination of the vortex towards the leading edge plane, as presented by Chen et al., is also observed. The influence of the increase of the loading on the total blockage in the flow passage was discussed by Tan [106]. In the case of a small clearance, a reduced flow coefficient induces mainly a growth of the suction surface boundary layer leading to the development of the corner separation. For larger clearances a high loading leads to an increase of the tip blockage related to the tip leakage flow.

In the compressor performance map given in the introduction to this section, Fig.2.1, the variation of the stability margin in function of the tip gap size indicates clearly an important influence of this flow on the compressor stability limit. The implication of the tip leakage flow and the related vortex in the compressor instability inception was observed and studied in a large number of publications (Wisler [116], Koch [64], Vo et al. [110], Day [26], Hoying et al. [56]). The results of these investigations are summarized by Tan [106]. The problem of compressor stability and the implication of the tip leakage vortex will be discussed in section 2.2.

2.1.2. Hub Corner Stall

The second important region of losses in a compressor is the corner between the suction surface of the blade and the end-wall at hub. This separation is present in every blade row. This region presents highly three-dimensional flows, generally with a separation inducing a pressure rise limitation, the creation of blockage and loss generation, *cfr* Cumpsty [22]. The presence of this three-dimensional flow is due to a velocity gradient in the viscous boundary layer.

In the main flow, a pressure gradient, parallel to the endwall and normal to the streamline direction, allows to follow the curvature of the blades. The amplitude of this cross-passage pressure gradient is sufficient to counterbalance the centrifugal forces w^2/R where w is the relative flow velocity and R the radius of the curvature. The free stream builds up this pressure gradient in the boundary layer where the flow velocity is lower than in the mainstream. This induces a reduction of the radius of the curvature followed by the flow in the boundary layer, as shown on Fig.2.7. It introduces a component of the velocity perpendicular to the local

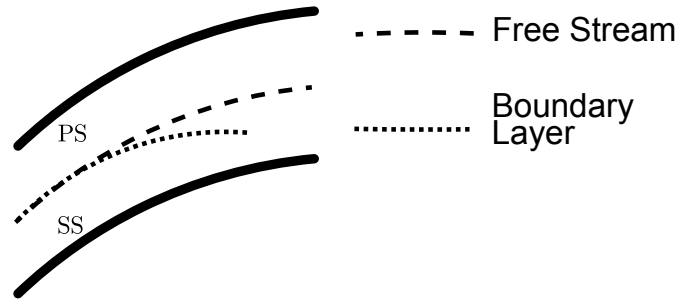


Figure 2.7.: Deviation of the boundary layer flow in the blade passage.

free stream. The boundary layer flow in the blade passage is deviated from the pressure side to the suction side. This secondary flow introduces an accumulation of a low momentum flow at the intersection of the suction side and the end wall, and this accumulated flow is further decelerated by the pressure rise. These two elements induce a separation of the flow both on the suction side of the blade and on the endwall, which leads to a reverse flow. This separation creates an important blockage of the blade passage which leads to a limitation of the loading and the pressure rise of the compressor. Furthermore the separation of the flow induces an important mixing with the main flow, inducing a decrease of the overall efficiency of the compressor. At high flow coefficients the reverse flow region on the endwall surface looks like a very thin layer along the blade suction surface, and the corresponding losses are not very significant. Once the blade loading exceeds a critical value, the reverse flow region expands abruptly in the blade passage, inducing an important increase of blockage and loss. The phenomenon is then

called hub-vortex stall. Figure 2.8 presents the corner separation flow topology without corner stall on the left hand side and with corner stall on the right hand side.

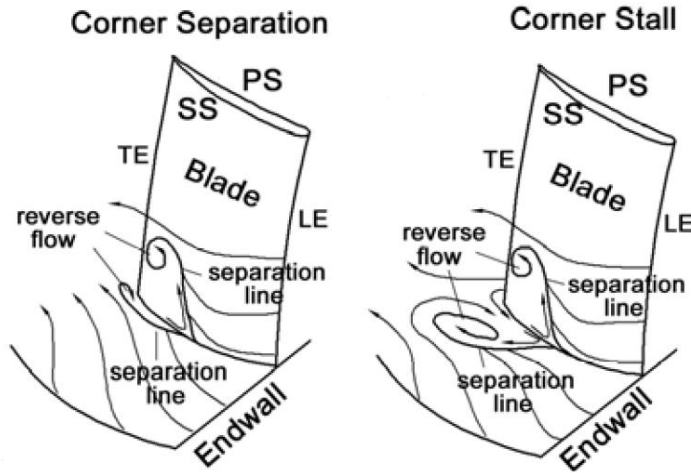


Figure 2.8.: Corner separation flow topology without (left) and with corner stall (right) [118].

The nature of the three-dimensional hub corner flow was experimentally investigated by Dring et al. [32] in 1982 on an isolated compressor rotor with high loading and low aspect ratio. The objective of this work was to provide a sufficiently detailed determination of the flow allowing a careful comparison with existing and future analytical models. The authors observed that the three-dimensional effects induced an important radial redistribution of fluid and that the proximity of the endwall had a strong influence on the blade pressure distribution. They showed, with flow visualizations, that the hub corner separation was highly dependent on the loading of the blade. A reduction of the flow coefficient leads to a radial increase of the extension of the separation area over the entire span. Dong et al. [31] presented a study on the flow field and loss mechanisms in a single-stage low-speed axial compressor. During these investigations they observed, with an oil flow visualization technique, a large separation zone at both endwalls of the stator but they did not find any at the rotor hub. The three-dimensional flow in an annular compressor cascade was experimentally investigated in 1988 by Schulz and Gallus [94]. They used extensive measurement techniques to perform a detailed flow characterization. The hub corner flow separation was observed at all investigated operating points. This separation and the secondary flows were the source of the high loss region observed downstream the row. Hah and Loellbach [44] (1999) detailed the flow structure of hub corner stall by using experimental

data and numerical solutions. They showed that the reverse flow on the blade suction side and the endwall is due to the strong particle motion around a vortex located at the corner of the hub and the blade suction surface. In 2005 Gbadebo et al. [40] were the first to show that a reverse flow region is always present both on the blade suction surface and on the endwall. Before that, the measurement techniques were not able to detect, on the endwall, the very thin layer composed by the reverse flow along the blade surface. That is why, before the work of Gbadebo, the hub corner separations with and without stall were distinguished by the presence of reverse flow on the endwall surface. They showed that the loss introduced by a hub corner separation, where the reverse flow is limited to the space close the blade, is relatively low in comparison with the case of hub corner stall. Gbadebo et al. conclude also from their experimental and numerical investigations that the presence of hub corner separation is universal. They validated the use of 3D Reynolds-Averaged Navier-Stokes codes for the prediction of the 3D separations by the comparison between measurements and numerical calculations.

Lei et al. [69] worked on the establishment of a criterion for the prediction of the hub corner stall in axial compressors. This work is based on the observation by Gbadebo of the limitation of the loss introduced by a hub corner separation in comparison with hub corner stall. Firstly Lei and al. defined a hub corner indicator S , quantifying the extent of the hub corner separation area by the calculation of the difference between the local blade loading inside and outside the separated region:

$$S = \Psi_{mid} - \Psi_{hub} \quad (2.1)$$

where *mid* and *hub* correspond respectively to mid-span and 0.1 span locations. Ψ is the Zweifel blade loading coefficient given by:

$$\Psi_z = \left[\int_0^1 \frac{P_{s,ps}(x) - P_{s,ss}(x)}{P_{t,up} - P_{s,up}} dx \right] \quad (2.2)$$

where *ps* and *ss* stand for blade pressure- and suction side, P and P_t the static- and total pressure, and *up* refers to upstream conditions. Next, Lei et al. established the criterion to predict the hub corner stall, the diffusion parameter D . The parameter D is a function of blade geometry and flow parameters. It was developed based on the hypothesis that the mechanism of hub corner stall is driven by the streamwise pressure rise, the cross-flow pressure rise and the skew of the incoming endwall boundary layer. The diffusion parameter D is then defined as a function of inlet- and outlet flow angles α_1 and α_2 , the incidence and blade camber angles i and θ , the flow turning due to boundary layer skew $\Delta\beta$ and the solidity σ as:

$$D = \left[1 - \left(\frac{\cos\alpha_1}{\cos\alpha_2} \right)^2 \right] \frac{i + \theta - \Delta\beta}{\sigma} \quad (2.3)$$

In order to determine the value of the diffusion parameter and the stall indicator corresponding to presence of the hub corner stall, Lei et al. calculated numerically

these two parameters for a very large set of geometries and established a correlation between them. The stall indicator in function of the diffusion parameter for these various geometries is plotted on Fig.2.9. Two distinct groups of results in S are

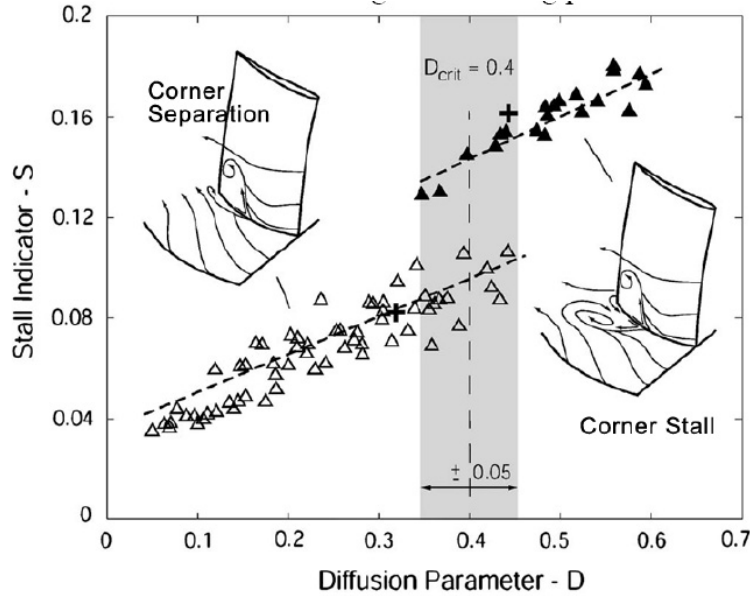


Figure 2.9.: Compressor stall indicator S in function of diffusion parameter D [69].

observable on this plot. All the geometries with a stall indicator greater than 0.12 present hub corner stall and only them. This value of the stall indicator is thus the critical limit for the occurrence of the hub corner stall. Concerning the diffusion parameter the critical value is equal to 0.4 with a tolerance of 0.05. This criterion being based on overall blade parameters, it can be used to determine the presence of hub corner stall during the preliminary design phase.

In 2010 Yu and Liu [118] proposed an improved version of the diffusion parameter established by Lei et al. [69]. The need for an improved parameter comes from the presence of a double valued region for S around $D = 0.4 \pm 0.05$ that can lead to a wrong estimation of the hub corner stall phenomenon, see Fig.2.9. Yu and Liu concluded from this observation that the influence of some factors were over- or under-estimated by Lei. The authors determined numerically that the blade aspect ratio, completely neglected by Lei, had a significant influence on the hub corner stall prevision. In the same way Yu and Liu showed that the stall indicator S defined by Lei was not adequate to predict the hub corner stall for blade aspect ratios lower than roughly 1. Thus, they defined in the first place a new stall indicator S_w equal to the maximum blade wake thickness non-dimensionalized by

the blade chord. Next they established the new criterion based on the observation that the diffusion parameter is: 1) affected proportionally by the circumferential and streamwise pressure gradient; 2) a function of the blade aspect ratio. Assuming a circular blade profile and the absence of significant flow separation, defining a simplified mean streamwise velocity in the blade passage and determining the effect of the aspect ratio on the diffusion parameter by a fitting method applied on the numerical data, the new diffusion parameter D_m can be expressed as:

$$D_m = \frac{(\cos\alpha_1 + \cos\alpha_2)^2 \sin\frac{\theta}{2}}{\sigma \cos^2\alpha_2} \left[1 - \left(\frac{\cos\alpha_1}{\cos\alpha_2} \right)^2 \right] AR^{0.18} \quad (2.4)$$

As in Lei’s work, the critical values of these new stall indicator and diffusion parameter were evaluated graphically. Figure 2.10 presents the stall indicator S_w in function of the modified diffusion parameter D_m for a very large set of data (about 200 blade geometries). The two branches of S_w , corresponding respectively to hub

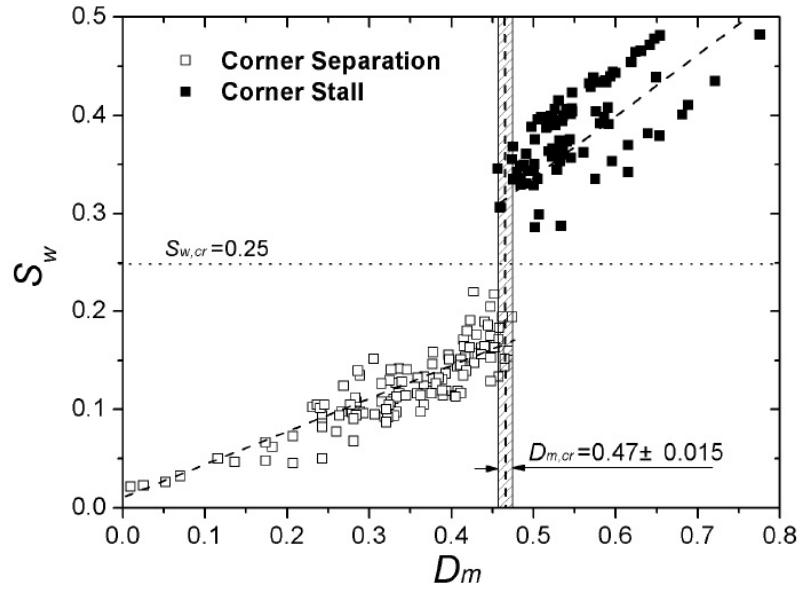


Figure 2.10.: Compressor stall indicator S_w in function of diffusion parameter D_M [118].

corner separation and stall, are clearly visible and a good separation criterion between them is given by a critical value of the stall indicator of 0.25. In terms of the modified diffusion parameter the limit between the two branches corresponds to a critical value of 0.47 with a relative uncertainty of 3.2% (± 0.015). This

new diffusion parameter was verified on compressor and cascade data found in the literature with a very good agreement.

In their works Lei [69] and Yu [118] established hub corner stall factors based on global flow parameters. They neglected completely the effect of the real geometry of the blades row. Goodhand and Miller [41] investigated the influence of small variations of the blade geometry on the development and extension of the three-dimensional separations. They considered various geometrical parameters like the leading edge geometry, the roughness and the blade fillet. The authors found that the fillet geometry on the blade or on the leading edge have a slight importance on the hub corner separations. On the other hand the geometry and the roughness of the leading edge between 5% and 30% affects strongly the extension of the hub corner separation and hence the hub loss via their influence on the transition process.

The role of hub corner separation on the stability of a compressor has not been studied extensively. In 2008 Choi et al. [18] performed a numerical investigation about its role on a low-speed axial compressor. They showed with time-accurate unsteady simulations that the stall cells were initiated from an initial asymmetric disturbance occurring at the hub corner separation. The initial disturbance moves from the hub to the tip region where it grows attached to the blade passage. Once the disturbance reaches a critical size it detaches from the blade and turns into rotating stall.

2.2. Compressor Stability Problems

The operating range of a compressor is limited on the high mass flow side by the choking limit and on the low mass flow side by the stability limit as shown on Fig.2.11. The choke regime corresponds to an operating point where the flow in the compressor reaches the sound velocity. This point corresponds to the maximum mass flow that can pass through the compressor. The stability limit corresponds to the lowest mass flow at which the compressor operates at a stable regime. A stable regime for a compressor is defined as an operating point where the compressor is capable to recover from disturbances. It is characterized by a steady periodic flow through the compressor. At an unstable operating point the flow through the compressor is unsteady and non-symmetric. All the stability limits at various rotational speeds make up the surge line. The instability of the compressor at low mass flow is induced by a flow separation on the blade due to a too high incidence angle. The process leading a compressor to reach its operating limit can be clarified with Fig.2.12. This figure shows the velocity triangle at the inlet of a rotor at design point and at a reduced mass flow operating point called throttled. At lower mass flow the absolute velocity is reduced from v_2 to v'_2 . Since the rotational speed of the rotor is constant, $U_2 = \dot{U}_2$, this leads to an increase of the relative inlet flow angle β_2 to β'_2 and thus to an increased loading of the blade. If the loading reaches a critical value, a flow separation occurs and the compressor works at an

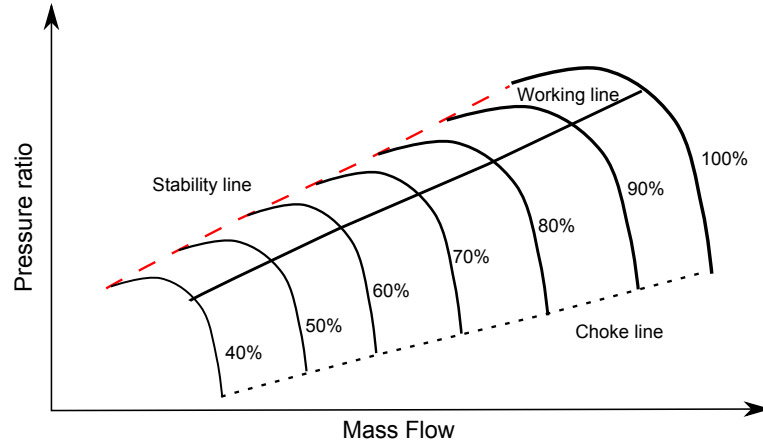


Figure 2.11.: Performance map.

unstable regime. The unstable regimes occurring in a compressor are the rotating stall and the surge. These regimes must be avoided in any case. Indeed they lead to a sudden drop of the compressor performance, inducing a loss of machine work and a mechanical shock in the machine, to very high vibrations on the rotor with a strong risk of blades destruction and, under some conditions, to a reverse flow through the blade row. This last point is particularly detrimental in the case of a turbomachine where the compressor is followed by a combustion chamber. Indeed the reverse flow induces the passage of hot gas from the combustion chamber through the upstream compressor. Compressor blades not being designed for high temperatures, reverse flow leads to the melting of these uncooled blades, causing an abrupt and definitive loss of the machine. An unstable regime in an aero-engine can lead to the crash of the airplane and thus mass casualties, especially if the instability occurs during landing or takeoff.

Due to the importance of operating in the stable region of a compressor, various stability approximation parameters have been defined. A first one was proposed by de Haller. This criterion is based on the limitation of the overall deceleration ratio in order to avoid a too high growth of the boundary layer. The de Haller criterion is given by:

$$\frac{w_3}{w_2} \geq 0.72 \quad (2.5)$$

where w_2 and w_3 are respectively the relative flow velocity at the rotor inlet and outlet.

Greitzer et al. [43] presented the non-dimensionalized static pressure rise across the rotor as a criterion linked to the casing boundary-layer. This criterion is related

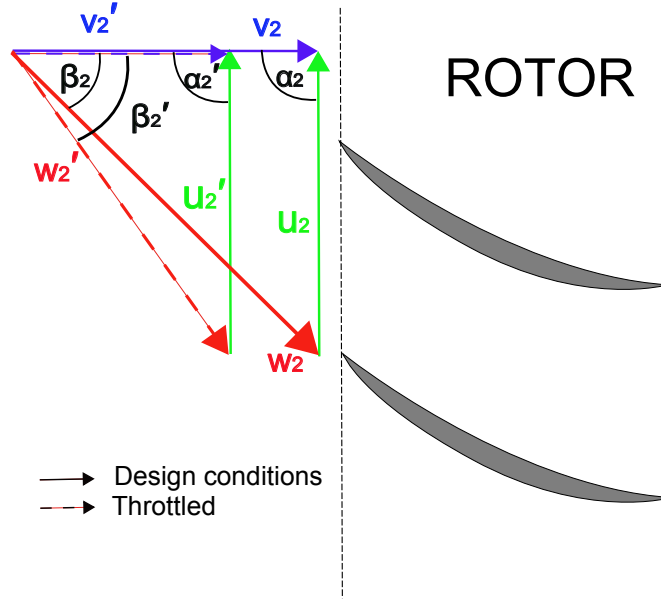


Figure 2.12.: Velocity triangle upstream the rotor for design and reduced mass flow conditions.

to the wall stall and given by:

$$\frac{\Delta P}{1/2 \rho w_2^2} \geq 0.6 \quad (2.6)$$

In 1975 Lieblein [73] showed the relation between instability and the diffusion factor D_f . The diffusion factor is based on the correlation between the flow deceleration along the blade suction surface and wake momentum thickness. It is given by:

$$D_f = \left(1 - \frac{w_3}{w_2}\right) + \frac{\Delta v_u}{2(\sigma) w_2} \quad (2.7)$$

and the stability limit is reached for $D_f \geq 0.6$. This stability criterion is associated with blade stall that will be described in the next section. It is important to notice the difference between Lieblein’s diffusion factor and the diffusion parameter presented in section 2.1.2. Diffusion factor and diffusion parameter consider both the static pressure rise as well as the flow turning. But the diffusion factor does not take into account the skewed character of the boundary layer due to moving end wall surface. This moving wall induces a deviation of the boundary layer flow in the circumferential direction. As presented by Lei et al. [69] the diffusion

parameter can be correlated with the diffusion factor only in the case without incoming end wall boundary layer skewness. The assumption that a compressor with a diffusion factor of 0.6 is stalled was contradicted by Cumpsty [22]. Indeed several examples show that a compressor with higher values of diffusion factor are still stable. This comes from the fact that a compressor presenting a stalled blade row can be stable. Cumpsty concludes that the diffusion factor is not a satisfactory parameter for the compressor stability.

The three parameters presented above give only an estimate of the stability character of a compressor. Actually, the stability of a compressor cannot be really predicted by any parameter. Experimental tests have to be performed to determine the exact stability limit.

The working line of a compressor is composed of the typical steady state operating points of the compressor at various rotational speeds as presented on Fig.2.11. The real operating point can be away from this line for many reasons, like inlet steady conditions (cruise or take-off in airplane engine), inlet unsteady phenomena or rotational speed changes. This last reason is critical for engines where strong accelerations are required. During a strong acceleration the compressor working point moves rapidly towards the stability limit and then slowly approaches the working line further up the map. This effect is due to the inertia of the compressor rows responding slowly to rapid changes of injected fuel mass flow. The increase of the fuel mass flow leads to a fast increase of the temperature and pressure in the combustion chamber, producing a high back pressure. On the other hand, the compressor rotational speed needs time to be adapted to the new conditions. During this time, the higher pressure ratio is not balanced by the higher rotational speed, leading to an increase of the blade angle of attack and a reduction of the stall margin. In case of deceleration the reverse effects are occurring. Consequently, the working line of a compressor must be placed far enough from the stability line in order to absolutely avoid an unstable regime. The difference in terms of mass flow between the working line and the stability limit is called the stall/surge margin, in function of the rotational speed of the machine (low-/high-speed) [26].

2.2.1. Rotating Stall

The rotating stall is a partial obstruction of a rotor rotating around the annulus, due to flow separation. The stall inception process and the time evolution of stall cells have been described in 1955 by Emmons et al. [35] and since then numerous authors have investigated this phenomenon. Actually rotating stall is not fully understood and its prediction remains difficult.

The stall inception is the process of compressor regime change between a stable operating point and an unstable regime. This transition period takes only a few rotor revolutions and presents two different processes. The first one is called the short length scale disturbance or spike stall inception. This process is triggered by a reduction of the flow rate in the compressor, entailing an increase of the incidence angle on the blades. Once the incidence angle reaches a critical value,

separation of the flow occurs on the suction side of a single blade near the casing, as presented on Fig.2.13. This separation leads to a blockage of one or two blade passages. The initial disturbance is thus size limited and measurements show that this disturbance is located in the tip region (Camp and Day [15]).

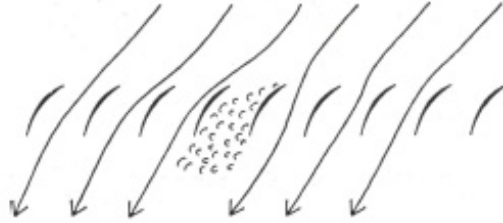


Figure 2.13.: Spike stall inception process [84].

The second stall inception model is characterized by long length scale disturbances, also known as modal perturbations. This model has been also presented in Emmons work mentioned above and precisely investigated by Moore and Greitzer [81] and Day [24]. In this model, it is assumed that the flow and the perturbations are two-dimensional, which means that they are uniform in the radial direction. In Fig.2.14 a mode of first order is presented. These modal waves start out as small disturbances. When approaching the stability limit they grow quickly in intensity without abrupt change in amplitude or frequency. At the end of the inception process a fully developed stall cell forms. Not only modes of first order are possible, also modes of higher order can occur. Contrary to spike inception, the modal waves are not a localized disturbance but are superimposed to the symmetric main flow over the entire circumference.

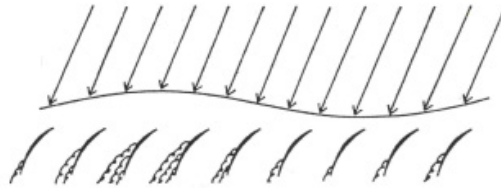


Figure 2.14.: Modal disturbance in a stall inception process [84].

Fig.2.15 presents the flow velocity measurements by means of hot-wires at six circumferential positions during spike and modal stall inception processes.

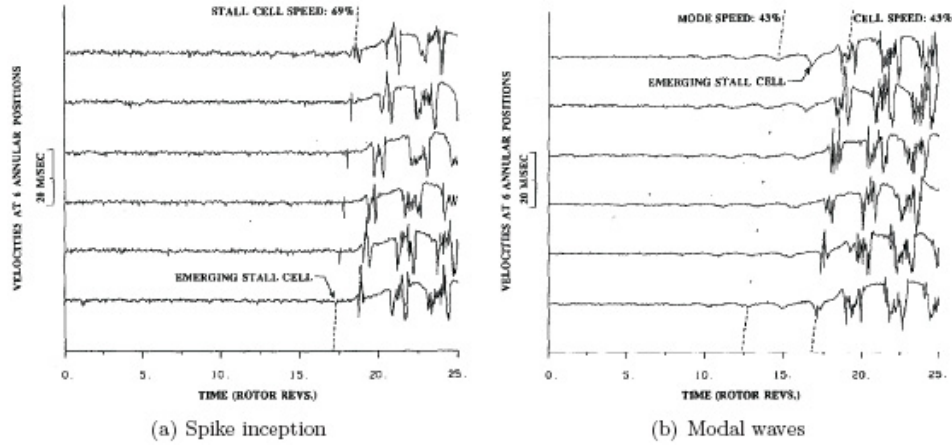


Figure 2.15.: Stall Inception [24].

Camp and Day [15] showed that the slope of the total-to-static characteristic at stall was linked to the stall inception mode. They observed that modal stall inception occurs in a compressor stage stalling at the peak of the characteristic. On the other hand, spike stall inception is typical for compressors where stall occurs before attaining the peak of the characteristic. The relation between the stall inception mode and the location of the stall point on the characteristic is explained by the inception process itself. The spike mode is caused by reaching the critical value of the incidence, while modular wave is due to two-dimensional perturbations and occurs at the zero slope point of the characteristic. If the compressor exceeds the critical value of the incidence before the peak of the overall characteristic, the spike appears, followed by a rotating stall. The compressor cannot then reach the peak of the characteristic and the modal waves cannot emerge [26].

Day [26] shows experimentally that the tip clearance has a crucial effect on the stall inception process. Furthermore, the origin of the spike initiating the rotating stall seems to be always in the tip region of the compressor. Ever since, many efforts have been made to identify the fundamental mechanisms in terms of tip leakage flow that trigger the rotating stall. A criterion for spike initiated rotating stall was formulated by Hoying et al. [56]. With his computations he could show that stall inception occurs when the vortex trajectory is aligned with the leading edge of the next blade passage. This assumption was confirmed and extended by Vo [110]. He formulated the following conjectures with the spike inception: (1) There must be a flow spillage from the tip clearance ahead of the leading edge of the next blade passage; (2) There must be a back flow of tip clearance fluid at the blade tip's trailing edge (Fig.2.16). The origin of the spike stall inception process is a separation in the tip region of the rotor blade caused by the tip leakage

vortex. This process is related to the wall stall, and compressors presenting this process are called tip-critical. Further experimental investigations concerning the inception process of rotating stall can be found in Bergner et al. [9], Mailach and Vogeler [79], Gannon et al. [39] and Lin et al. [76].

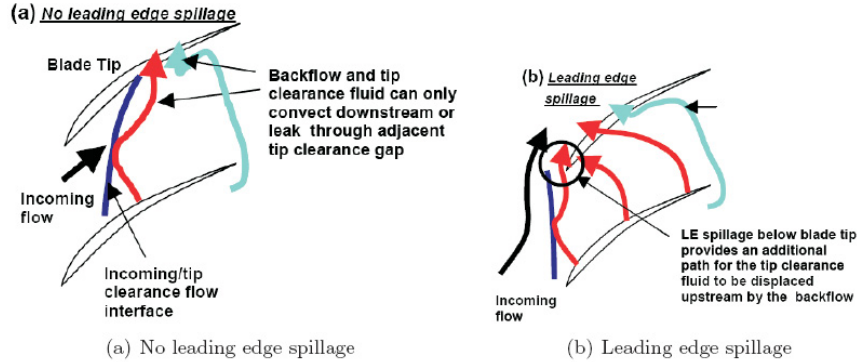


Figure 2.16.: Criteria for spike inception [110].

Once the stall cells are fully developed, following one of the processes presented above, the cells are circulating around the annulus. Indeed the blockage of a blade passage induces a deflection of the flow to the following blade passages, causing an increase of the incidence on them and thus a separation of the flow in this region. At the same time the blockage leads to a decrease of the incidence of the previous blade passage, causing a reattachment of the flow. These two effects cause a displacement of the stalled region of the compressor around the annulus in opposite direction of the rotation of the rotor. Due to the lower rotational speed of the stall cells relative to the rotor speed, the cells are rotating around the annulus in the same direction as the rotor in the absolute reference system.

The fully developed rotating stall can be differentiated in two general types: 1) the part-span stall, where only one extremity of the blade is affected; 2) the full-span stall, where the complete height of the blade is affected. Day [26] describes some distinguishing features of the two types:

Part-span stall cells

- Part-span stall cells usually occur in compressors where blade aspect ratios are large,
- The number of stall cells is nearly always greater than 1,
- Part-span stall cells will always be equally spaced around the annulus,
- The stall cells usually only affect the tip region of the blades where their presence represents a net blockage in terms of through flow area,

2.2. Compressor Stability Problems

31

- Part-span stall cells usually rotate at about 50% to 80% of the rotor speed,
- Part-span stall cells are often confined to just one or two blade rows at the front of a compressor and seldom stretch all the way through a multi-stage machine,
- The relative small pressure loss caused by part-span stall cells means that most aero-engines can tolerate the presence of these stall cells at low engine speeds.

Full-span stall cells

- Full-span stall cells occur in most high-pressure compressors, especially in multistage machines with short blades,
- Full-span stall cells cover the full height of the blade and leave no room for flow to pass under the stall cell at the hub,
- Full-span stall cells usually give rise to a single stall cell, particularly in multistage machines. Multiple full-span stall cells are very seldom seen,
- In the rare cases where more than one full-span stall cell develops, the stall cells will not necessarily be equally spaced or of equal size,
- Full-span stall cells rotate more slowly than part-span stall cells, usually between 20% and 50% of the rotor speed,
- The dramatic loss of pressure which occurs when a full-span stall cell forms, means that most aero-engines are forced to shut down if stalling occurs. Acceleration of the engine during full-span stall is not possible.

In 1978, Day and Cumpsty [28] established the existence of a relation between the size of the stall cell and its type. From the analysis of data from various experimental study of compressor stall phenomenon, they suggested that stall cells type can be predicted from the fraction of the annulus covered by the cells λ . The authors determined that 30 % was the critical value of λ , separating the full-span and the part-span stall cells. According to Day and Cumpsty, 30 % is the maximum fraction of the annulus covered by part-span stall cells, and the minimum for full-span stall cells.

The two types of stall cell can occur in the same compressor, as presented by Mathioudakis [80]. In this work, the author observed two different unstable regimes in a compressor. First, the instability presented multiple part-span stall cells and led to a progressive deterioration of the compressor performance. Then, a second mode of instability occurred, with one or two cells and characterized by an abrupt performance drop.

2.2.2. Surge

The surge is an unsteady phenomenon oscillating between a normal operating regime and a total obstruction of the compressor. For many years the difference between rotating stall and surge was not well defined. In 1978 Greitzer [42] told the distinction between the two phenomena. In this paper Greitzer also established the non-dimensional Greitzer B parameter, giving a way to predict if a specific compressor would surge or stall. This parameter was extended by Day [25], providing the possibility to compare different compressors. Recent developments of the knowledge of the surge phenomenon have confirmed that surge in axial compressors is initiated by rotating stall.

The surge phenomenon is shown on the right hand side of Fig.2.17. At point 1 the compressor in stable regime reaches the stability limit, and rotating stall occurs. This causes an abrupt loss of pressure rise and the loss of the ability of the compressor to support the high pressure energy stored downstream, causing a reverse flow, point 2. This reverse flow induces a reduction of the downstream pressure and, at point 3, this pressure is low enough to un-throttle the compressor. Then the compressor regains a normal flow (point 4) until the stability limit is reached again. The compressor follows a repetitive pumping cycle. This process can only be interrupted by changing the compressor’s conditions like decreasing the rotational speed or the throttling.

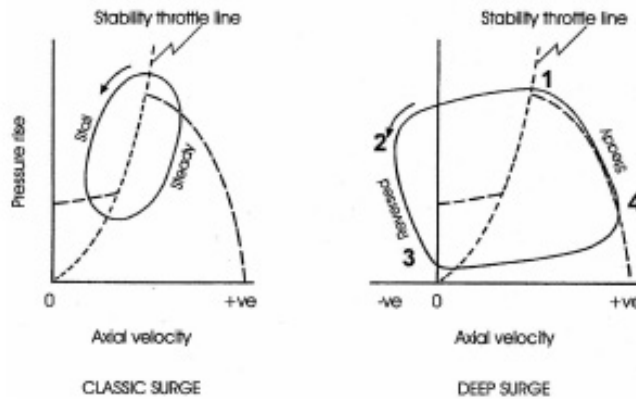


Figure 2.17.: Classical (left) and deep (right) surge phenomena [42].

Surge can be classified in two main types. The “deep surge” with reverse flow described above, and the “classic surge”, where no reverse flow occurs. In Fig.2.17 the two types are presented. In a classic surge cycle, the energy stored behind the compressor is not sufficient to cause a reverse flow. The compressor changes its

conditions from stalled to un-stalled and the flow rate changes between fast and slow.

2.3. The Casing Treatments

As mentioned in the previous chapters, the stall and surge phenomena have a high destructive potential and must be avoided at all cost. The tip leakage vortex plays an important role concerning the stability limit of a compressor. Over the last 40 years various mechanisms have been investigated to stabilize the flow in the tip section and thus extend the working range of the compressor. These mechanisms can be categorized as active or passive means. The active control mechanisms are based on a real-time stall detection and devices allowing getting the compressor out of stall. These devices are for example bleed valves, injectors, variable stagger-angle stators,... Several investigations shown the potential of these mechanisms on the improvement of stall margin [93] [104] [36] [103]. The passive systems are based on the modification of the geometry of the compressor. The most investigated passive method in recent years has been the usage of casing treatments (CT). A casing treatment is a geometrical modification of the inner surface of the compressor casing. Various kinds of arrangements like circumferential grooves, axial slots or honeycombs exist. The main principle of the casing treatment is to extend the working range of turbo-compressors by decreasing the amount of blockage in the flow passage. The stabilization potential of passive casing treatments has been demonstrated through a large number of studies.

2.3.1. History of Casing Treatments

The concept of working on the endwall flow in order to enhance the stability of a compressor has appeared in the 1950's. The first document concerning the increase of the compressor stability based on end wall flow improvement is a patent of Wilde [112]. The patent describes a bleeding system where the air was sucked in front of the Inlet Guide Vane (IGV) and bleeds it along the casing ahead the IGV. In the early 1960's the NASA commissioned a series of investigations of casing-blowing and -bleeding on high-speed single stage axial compressors. The tests showed good results concerning the surge margin for the blowing configuration.

In 1970, Bailey and Voit [6] developed and investigated the first passive endwall treatments. They used tapered holes, honeycombs and radial holes connected to a plenum as casing treatments, without external bleeding or blowing. Furthermore, the effect of axial and circumferential flow recirculation was investigated by using baffles. The configuration with tapered holes and honeycomb gives a significant improvement of the stability range. The variation of the stability improvement by the tapered holes in function of the arrangement of the baffles shows that the pressure rise induced by the rotor can be used to improve the stability range by allowing a flow recirculation and a natural reinjection ahead of the rotor tip.

Motivated by the ability of casing treatments to stabilize the end wall flow, a large amount of parametric investigations on CT was performed in the 1970's as extensively reviewed by Hathaway [48], Fig.2.18. Various configurations like honeycomb, radial holes, axial slots, skewed slots, blade angled slots and circumferential grooves were considered. Furthermore the positive effect of CT without plenum on the end wall flow was demonstrated. These investigations did allow a better understanding of the flow mechanisms occurring in a casing treatment. Most of the configurations tested provided a stability range extension especially

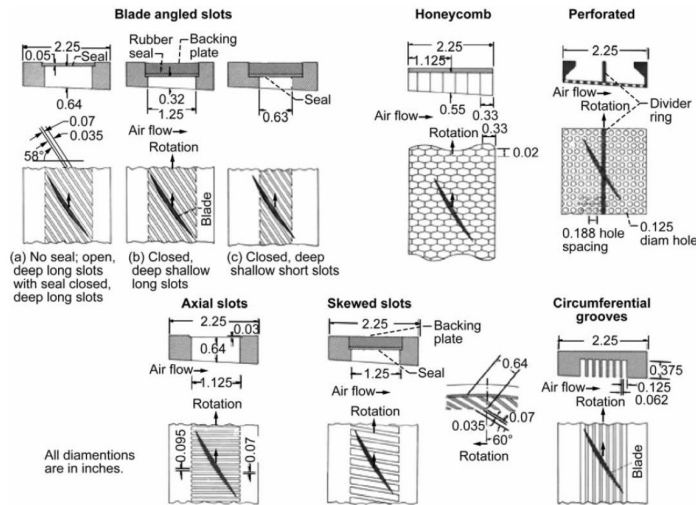


Figure 2.18.: Casing treatment configurations investigated in the 1970's [48].

in case of tip-critical compressors. Osborn et al. [83] showed that the higher stabilizing potential of the skewed slots, in comparison to circumferential grooves, came with an unacceptable loss of efficiency. In the case of grooves the maximum efficiency was as high as, or slightly higher than, the one obtained with solid wall but the effectiveness of the CT was dependent on the rotational speed. Osborn et al. insist on the choice of casing treatment in function of the compressor. These results were confirmed by the work of Moore et al. [82] and Bailey [5].

Due to problems with the physical understanding of the interaction between tip leakage flow and the flow in the casing treatments, many inconsistencies were observed in the investigations of the 1970s. In their work Prince et al. [88] focus on the comprehension of the physical effect of CT. This work is based on a survey of former investigations and on personal extensive investigations of several casing treatments including hot-wire measurements in treatment cavities. They were able to show that compressible effects and acoustic resonance were not the crucial effect of the positive influence of CT. However, they were not able to provide a better

understanding of the working mechanism of CT.

Based on the works of Osborn et al. [83], Moore et al. [82] and Bailey [5] showing skewed slots providing the greatest range extension, Takata and Tsukuda [105] investigated the importance of the orientation of the slots on the performance of the CT. They showed that the configuration where the slots are skewed opposite to the rotor rotation direction gives the best results in terms of stability improvement. They explained this fact by the circulation of the flow in the slots and the pumping mechanism into the slots. They did also show that improper orientation can lead to a deterioration of the stability of the compressor. By the use of hot-wire measurements in the slots cavities, they showed a periodic system of pumping into the slots near the trailing edge and expulsion from the slots to the main flow upstream the mid-chord. Takata and Tsukuda hypothesized that the main mechanism of stability margin improvement by the use of skewed slots casing treatment is the exchange of momentum, facilitated by the dynamic effect of the expelled jet.

In 1979, Greitzer et al. [43] made a systematic experimental investigation of the influence of axial skewed grooves on the stability margin. The investigation was performed on two rotors presenting different solidities. They observed and described a different stall mode for each compressor: 1) The blade stall, where an aerodynamic flow breakdown occurs on a significant portion of the blade span; 2) The wall stall where the breakdown begins in the tip region, due to the low momentum of fluid in the boundary layer. The two stall modes are presented on Fig.2.19. Casing treatment was observed to influence the working range of only the compressor exhibiting a wall stall. These results have been the initial point of investigation for a better understanding of the flow phenomena that cause the positive effects on the stability range and the partially negative effects concerning the efficiency of compressors.

The investigation on the relation between the main flow inside the passage and the flow in the casing treatment was performed by Smith and Cumpsty [99]. They used hot-wire measurements in axial slots to determine the velocity vectors in the casing treatment slots. It can be observed that a mass flow enters the treatment at the trailing edge of the blades and is re-injected at the leading edge. These results lead to the assumption that the stabilizing effect of casing treatments does not occur in the CT itself but rather in the mixing area generated by the impingement of the re-injected flow on the main flow outside the casing treatment, as conjectured by Takata and Tsukuda [105].

During the same year Fujita and Takata [38] made a study on the determination of the optimum between a high improvement of stall margin and the lower efficiency loss induced by the use of a casing treatment. This study is based on a large number of casing configurations like axial, skewed and circumferential grooves. They found a linear relation between the two parameters leading to the assumption that an increasing stability range always comes along with a decreasing efficiency.

With Crook et al. [21], in 1993, the numerical investigation of CT began. With a steady three dimensional Navier-Stokes solver they investigated the flow field

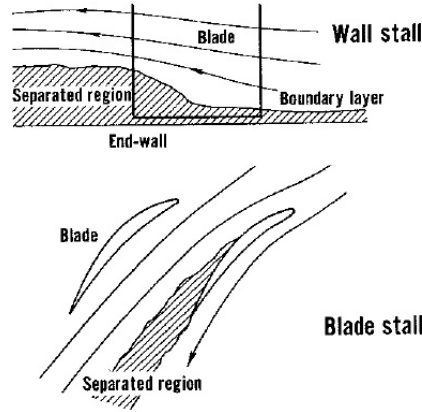


Figure 2.19.: Compressor stall inception mode [43].

in a compressor with axial slots as hub treatment. An extensive investigation of circumferential grooves and skewed slots was made by Hall et al. [45] and [46]. They made steady and unsteady simulations with and without inlet distortion on various compressor- and fan stages. More recent numerical simulations on casing treatments were performed by *e.g.* Wilke and Kau [115], Brignole et al. [10] and Shabbir and Adamczyk [95].

Confirmed by the amount of numerical and experimental investigations, performed in the last 40 years, the stabilizing mechanism of casing treatments has been demonstrated. This applies to low speed machines like fans or blowers as well as to high speed machines. Although the complex physical phenomena that underly the positive impact on the tip clearance flow are not completely understood so far, several types of CT are actually in use. To quantify the effectiveness of a casing treatment, five criteria have to be taken into account:

- Improvement of the stability margin,
- Negligible negative impact on the compressor efficiency,
- Gain in pressure ratio,
- Ease of manufacturing and assembling,
- Minimal space requirements,

Furthermore, considering the efficiency of a real turbo-compressor, the working line does not always correspond to the maximum efficiency because of the stability margin requirements. An extension of this margin with casing treatment may allow a shift of the working line to the maximum efficiency.

2.3.2. Circumferential Groove Casing Treatment

A circumferential groove casing treatment generates smaller improvements in terms of stability margin but causes a smaller efficiency drop. For lots of applications this small margin increase is therefore sufficient and the very limited negative influence of the CT on the efficiency is an essential requirement. Furthermore the simplicity of manufacturing and assembling of this type of casing treatment makes it very often implemented.

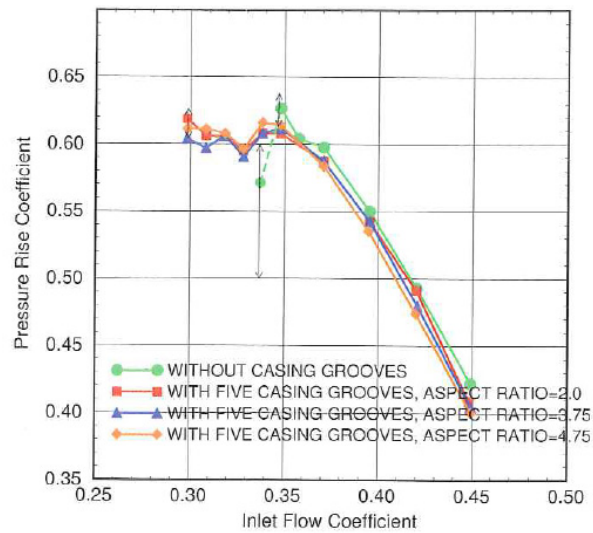


Figure 2.20.: Pressure rise coefficient of a low speed axial compressor with and without casing treatment [96].

The stability range extension of a compressor by using circumferential grooves casing treatment is due to an interaction between the tip leakage vortex and the grooves. The working principle is a migration of the fluid from a severe blade loading and high pressure region to a location where the pressure is more moderate as presented by Beheshti et al. [8]. This mechanism provides a local decrease of a critical loading on the blade and thus has a stabilizing effect.

In 2005, Shabbir and Adamczyk [96] performed a numerical investigation on the effect of circumferential grooves on a low speed axial compressor. This study improves the understanding of the physical mechanism causing the stability improvement by the use of this type of casing treatment. The overall performance of the compressor considered by Shabbir and Adamczyk for various casing configurations is presented on Fig.2.20. The compressor stability range looks insensitive to the grooves aspect ratio. In this work the authors wanted to demonstrate the

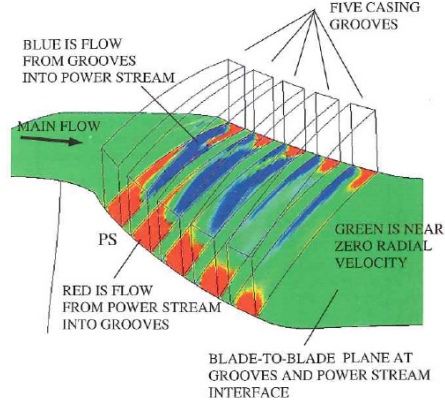


Figure 2.21.: Radial velocity distribution in the casing plane [96].

decrease of the growth rate of flow blockage by the use of a circumferential grooves casing treatment. Shabbir and Adamczyk performed the budget analysis of the axial momentum equations for a control volume adjacent to the casing for solid casing configurations and casing treatment configurations. The control volume is radially bounded by the end wall and the first grid cell from the wall, according to the hypothesis that the growth rate is “controlled by the near casing flow field over the tip”. From the results, they found that, in the case of a grooved casing,

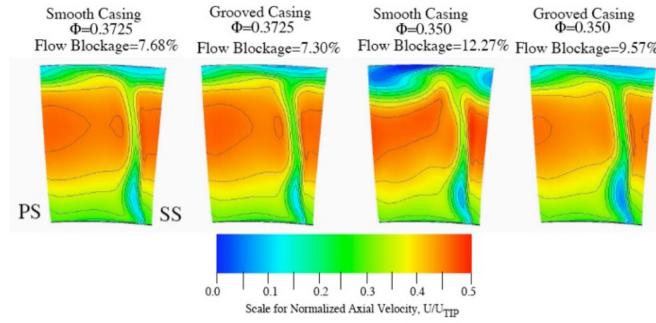


Figure 2.22.: Normalized axial velocity contours downstream the rotor trailing edge for two casing configuration and two flow conditions corresponding to conditions away from stall ($\phi = 0.3725$) and close to stall ($\phi = 0.35$) [96].

the net axial pressure force at the casing in the tip gap is compensated by the net

axial force due to the radial transport of axial momentum across the grooves, see Fig.2.21. They concluded that this radial transport is the flow stabilizing mechanism. Finally Shabbir and Adamczyk determined the flow blockage in the blade passage by using the definition proposed by Khalid et al. [61] for the various configurations. This approach is based on the axial velocity just downstream the rotor trailing edge plane. Under conditions away from the stability limit of the compressor, the difference of the flow blockage for the smooth casing and with the casing treatment is less than 0.4% around 7.5% and can be neglected. Close to the stall point the flow blockage in the smooth casing configuration grows to 12.3% and only to 9.6% in the case with grooves. The flow blockage in the various cases is presented on Fig.2.22 at the normalized axial velocity. These results show the rapid flow blockage increase, corresponding to a reduction of inlet flow coefficient. The hypothesis of the authors about the reduction of the blockage is then validated.

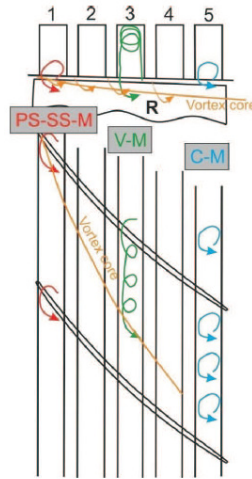


Figure 2.23.: Flow mechanism in circumferential grooves [50].

In 2008 Hembera et al. [50] made a systematic comparison of investigations performed until then. They concluded to the following:

- Different groove depths have different effects on the efficiency and in shifting the surge line, depending also on the number and axial position of the grooves,
- In most cases, more grooves have a higher potential in shifting the stall margin but also show a greater loss in efficiency,
- Altering the inner geometry of the grooves, for example by including baffles

that separate the circumferential grooves in segments, influences the shifting of the stall margin,

- The effects of the grooves on the compressor stage is strongly dependent on the axial location of each groove relative to the other grooves and to the blade,
- The inner geometry of the grooves, for example their individual depths which can be different, are at least as important as the axial position above the rotor or the number of grooves in terms of efficiency change and stall margin improvement.

Moreover, they performed a numerical investigation, using one to five casing grooves in order to analyze the flow mechanism in various axial positions relative to the rotor chord. The flow mechanism for the five grooves is presented on Fig.2.23. In the first two grooves, placed at the leading edge, the main mechanism driving the flow is the high pressure difference between the suction- and the pressure side of the blade. The mechanism that affects the flow in the center grooves is the vortex mechanism, caused by the static pressure in front of the blade. At the trailing edge, the cavity mechanism is dominant. It is the weakest mechanism and the flow is nearly perpendicular to the front side of the groove.

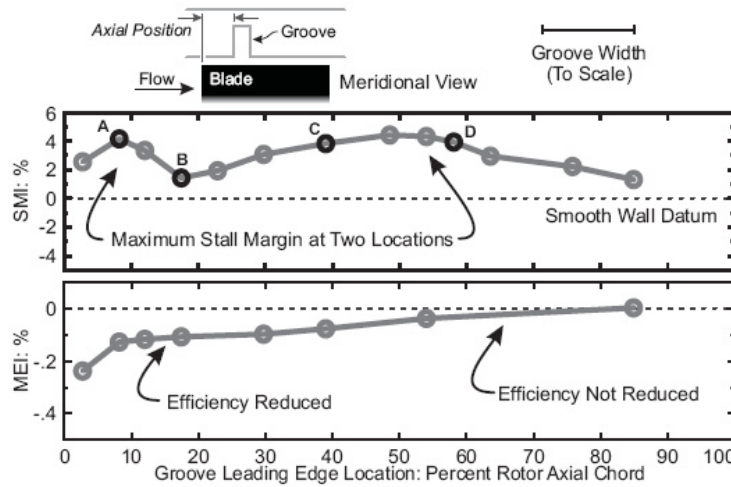
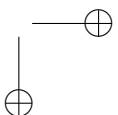
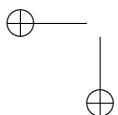
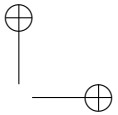
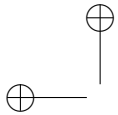


Figure 2.24.: Stall margin (SMI) and maximum efficiency (MEI) improvement in function of the groove casing treatment axial location [53].

Houghton and Day [53] investigated the influence of the axial location of a circumferential groove casing treatment on the stability and the efficiency of a low-speed axial flow compressor. Furthermore they detailed the interaction of

the groove on the flow in the tip region. Two optimal locations for stall margin improvement were found at 8 and 50% of the axial chord, Fig.2.24. At 8% the strong interaction of the groove on the highly energetic flow field near the casing causes high losses and hence an efficiency reduction. At 50% the groove intensifies the tangential velocity of the flow in the casing region without perturbing the flow field, thus inducing negligible losses. In this study Houghton and Day demonstrated the absence of a direct link between flow passage blockage and stall margin improvement.

In 2010 the same authors [54] showed that the efficiency loss due to multiple grooves is equal to the sum of the drops from each single groove while the stall margin improvement is less than the sum of the improvement due to each single groove. The importance of the stall inception mode on the effectiveness of the casing treatment is also discussed. This effectiveness is found to be maximum for compressors presenting spike stall inception for both casing configurations. However, the effect is decreased if the stall inception goes over from spike mode in a smooth casing configuration to modal mode with casing treatment. The effectiveness is very low for compressors exhibiting modal stall inception for both casing configurations.



Chapter 3.

Experimental and Numerical Tools

This thesis presents the experimental characterization of a single stage compressor representative of the first stage of the booster designed for the new contra-rotating turbofan architecture. This characterization has been performed on the R4 high speed compressor facility of the von Karman Institute. The measurements are compared with the results of numerical simulations. This allows validating the ability of the numerical code to simulate the global performance and the stability limit at high loading- and flow coefficients, and of low aspect ratio, and that with various CFD codes and turbulence models. Once the simulations are validated, the numerical results are used for the flow visualization in the compressor stage.

The experimental and numerical tools used in the frame of this work are presented in this chapter. Firstly, the characteristics of the R4 facility used for the compressor characterization are presented. Then, the single stage compressor test section is presented. This presentation includes the design, the instrumentation layout and the displacement systems for the instrumentation. The details of the aero-dynamic design of the blades of this particular compressor are described.

The details of the numerical simulations, used in the comparisons, are given. These details concern the various codes and turbulence models used for the simulations as well as the informations about the meshes considered. The numerical results presented in this work come from the work of Balazs Farkas, PhD at the VKI, and from CENEARO, a partner in the VITAL consortium.

Finally, the tests conditions and configurations are presented. The compressor stage has been investigated at two Reynolds numbers, corresponding to the cruise and take-off conditions. The parameters used to reach these Reynolds numbers are given. Furthermore, the effect of a circumferential groove casing treatment on the compressor performance was investigated as well. The geometrical characteristics of this groove studied during this thesis are detailed.

3.1. The VKI R4 Test Bench

The R4 test facility is a high speed compressor test rig with a closed loop system. This facility was designed and built in the late 60’s to study high speed axial and radial compressors. After being idle for twenty years, the Turbomachinery Department decided in 2004 to refurbish and modernize the R4 test rig for the current VITAL project and others to follow.

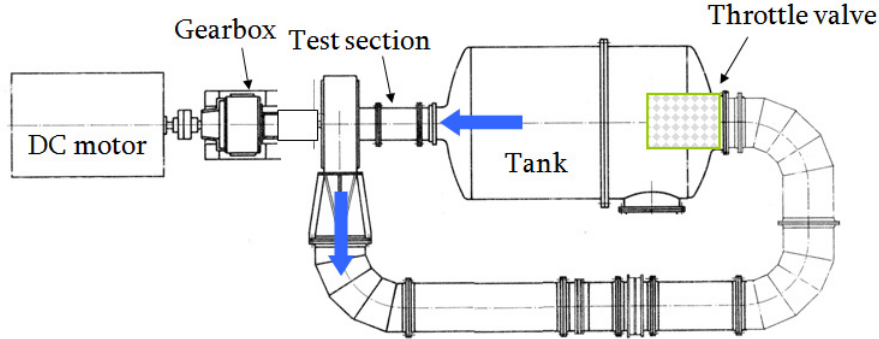


Figure 3.1.: Layout of the VKI-R4 closed loop high-speed compressor test rig

This rig is composed of a tank used as settling chamber followed by the test section and the diffuser where the flow is decelerated into a piping leading the air back to the tank. The settling chamber is equipped with an air/water heat exchanger, allowing the control of the flow temperature in the test section inlet, and a throttling valve providing the means to run the compressor at various operating conditions. For axial compressors, the R4 facility limits the diameter of the compressor hosted by the test section at maximum 480mm . Figure 3.1 shows the test bench layout. The closed loop system allows controlling the pressure inside the facility from 300mbar absolute to a maximum of 3bars absolute. This allows managing the Reynolds number independently of the rotational speed of the rotor and thus independently of the Mach number. This gives the possibility to determine the characteristics of the compressor under conditions corresponding to cruise (CR) or take-off (TO) conditions.

By the fact of the facility being out of use for twenty years, its stripping down and the storage of some parts in the open, an important work of refurbishing was necessary. Furthermore, various elements of this old facility required a modernization in order to keep up with current standards. The refurbishing and modernization of the installation included:

- The modification and the motorization of the throttle valve,
- The modification and the improvement of the cooling system,
- The modification of the back-loop with the addition of expansion join,
- The addition of a torque-meter on the shaft,
- The modification and the modernization of the oil system,

- The re-alignment of the motor, of the gear-box, of the test section and of the tank.

The rotor is driven by a $695kW$ DC motor with a nominal speed rating of $1000RPM$ at $1000V$ supply and is coupled to a 12:1 gearbox. However, the voltage supply is currently limited to $750V$, which limits the compressor speed to $9000RPM$. The diagram of available compressor power and the rotational speed is shown on Fig.3.2. The rotational speed of the rotor and the exact power given by the motor to the compressor via the shaft are measured by a torquemeter. The torquemeter was chosen to cover the entire range of the electrical motor and has a resolution for the rotational speed of $1RPM$ and an accuracy for, respectively, the torque and the speed better than 0.1% and 0.04% .

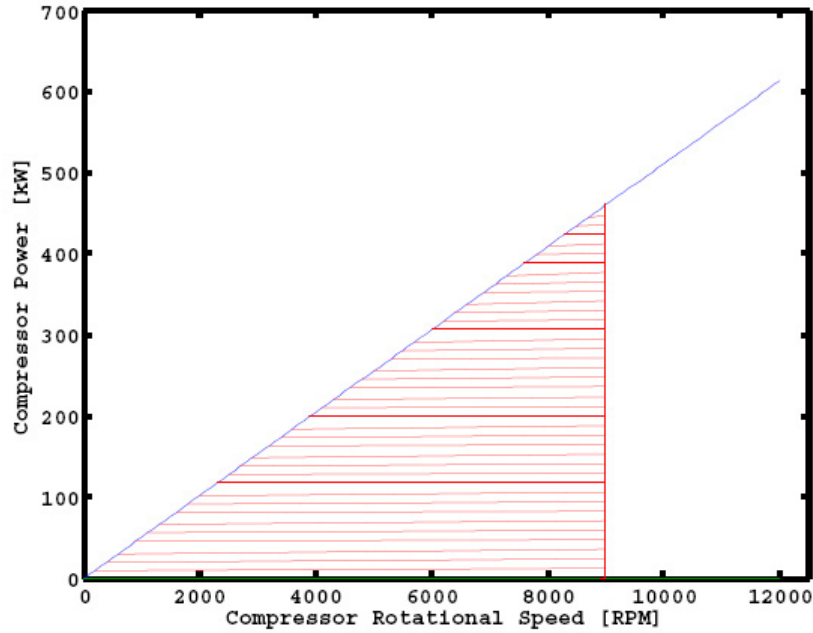


Figure 3.2.: Diagram of the available compressor power in function of the rotational speed of the R4 compressor test rig.

The loop was initially equipped with a diaphragm used for the determination of the mass flow passing through the compressor. But, considering the low pressure ratio of the stage, the pressure drop induced by the diaphragm at low speed was too large for the compressor to get out of stall. Furthermore, there was not the required pipe lengths up- and downstream the diaphragm for the use of the international standard equations ([1]). This procedure was then replaced by a mass

flow determination based on direct measurements of the pressure, temperature and flow velocity at the inlet of the test section.

3.2. The VITAL Test Section

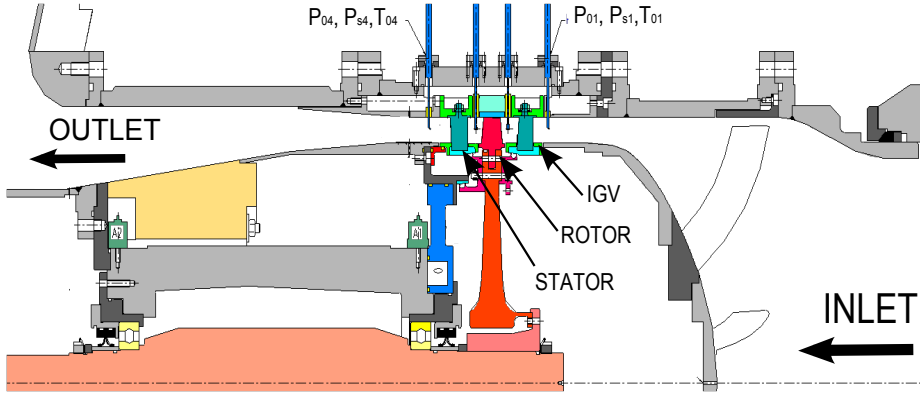


Figure 3.3.: Layout of the VITAL test section

Due to the diameter limitation of the test facility, the compressor stage was designed with a scale of 0.5 compared to the real first stage of the booster. The cross-sectional view of the test section is presented on Fig.3.3. This test section is divided in three parts. The first part is comprised between the inlet of the test section and the IGV. In this part, the flow is led from the outlet of the settling chamber to the blade rows. A bulb allows a smooth reduction of the test section and, then, an acceleration of the flow. This bulb is maintained by six arms to the tip casing. The bulb and the arms are visible on the picture of the test section presented on Fig.3.4. These arms and their positions are designed in order to reduce their influence on the flow in the compressor stage.

The second part represents the compressor stage itself. The stage is composed of the IGV, the rotor and the stator. Adding the IGV to the stage is required in order to maintain the similitude of the flow conditions at the inlet of the rotor. Indeed, in the real engine, the inlet flow to the booster is nearly axial due to the realignment of the flow direction by the downstream fan. But the inlet total pressure and axial velocity distributions are highly skewed due to the non-uniform pressure rise in the hub region of both fans. On the contrary, the test rig compressor sees a pure axial and uniform inlet flow. This leads to the introduction of inlet guide vanes (IGV) to the stage to match the rotor relative inlet angle distribution, and hence the correct loading distribution. Nevertheless, the axial velocity gradient is not matched, inducing vorticity in the main flow. The real stage has variable radius

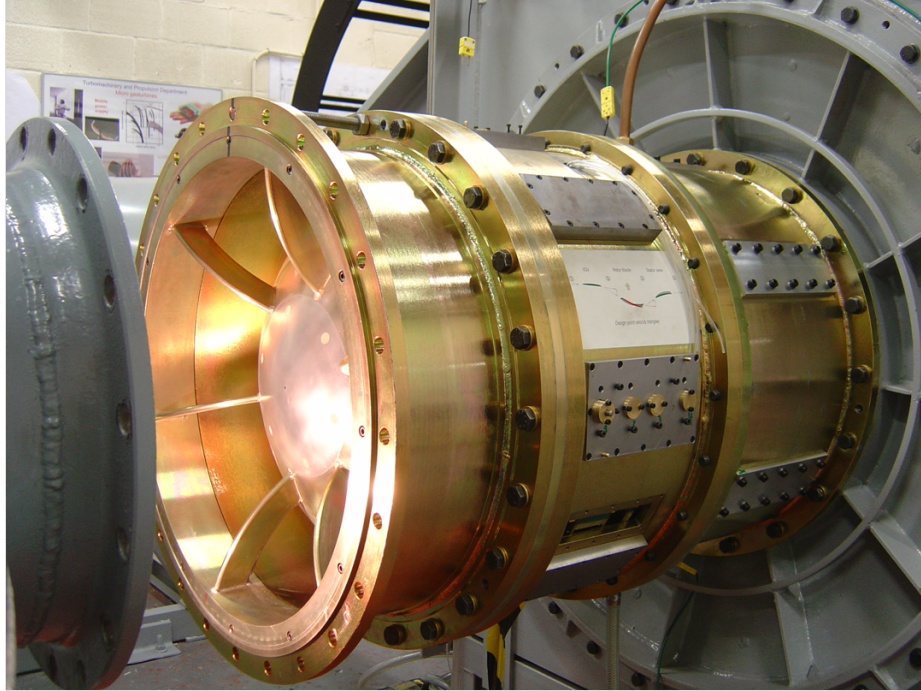


Figure 3.4.: Photograph of the inlet of the VITAL test section

hub and tip contours whereas the meridional flow path of the test compressor stage was designed with a flat tip contour for the ease of manufacturing, and also for the control of tip clearance and in view of performing experiments on various casing treatments. The hub contour was designed to preserve the area contraction. The three blades rows are mounted on separate platforms allowing the rotation of the compressor rotor. In order to avoid any blockage and important flow perturbation due to positive step between two plate-forms, a negative step of 0.1mm has been imposed in the design at each blade row platform interface. The geometrical characteristics of the compressor stage are given in table 3.1.

The actual rotor tip clearance was measured by wear gauges at 6 different circumferential positions both at leading edge and trailing edge. The design aimed at a nominal clearance of 0.3mm (1.3% of blade height) and was achieved at the commissioning. The actual tip clearance under running conditions at nominal speed (100% NN) was found to be between 0.12 and 0.25mm , i.e. between 0.5% and 1.08% of the blade height. The corresponding distribution is shown in Fig.3.5 that indicates also a slight excentricity of the rotor axis to the casing. The second set of curves indicates the evolution of clearance obtained after running the compressor

Number of stages	1.5
Number of IGV/rotor/stator blades	87/80/87
Rotor tip radius	240 mm
Rotor blade height	23.04 mm
Rotor chord at hub	23.66 mm
Hub to tip ratio (rotor inlet)	0.9
Design tip clearance	0.3 mm
Average solidity (Rotor/Stator)	1.23/1.17
Aspect ratio (Rotor/Stator)	1.18/1.20

Table 3.1.: Stage characteristics of the VITAL test section

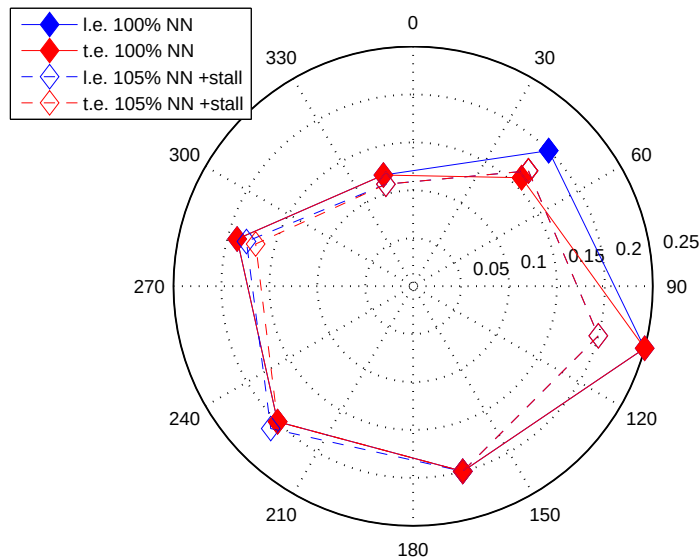


Figure 3.5.: Circumferential distribution of the rotor tip clearance at leading and trailing edges at 100% and 105% nominal speed and after stall

into stall at nominal speed and up to 105% of nominal speed (105% NN + Stall). After this test, only a very slight change in clearance could be noticed, except between 90° and 120°, where the tip gap was reduced from 0.25 to 0.2mm. Of course it has to be recalled that the wear gauge technique is providing a measurement of the smallest gap as created by the longest blade on the rotor.

In order to get the complete flow characteristics in the compressor, various aerodynamic parameters (static and total pressure, total temperature, flow angle,

Mach number) have to be measured at four axial positions in the compressor stage. These positions correspond to the stage inlet (station 1), the rotor inlet (station 2), the rotor outlet (station 3) and stage outlet (station 4), as shown on Fig.3.6. Usually, the stage performance is measured by inlet and outlet rakes. This option was not used in the present case, considering the very small blade height and the blockage induced by the rakes.

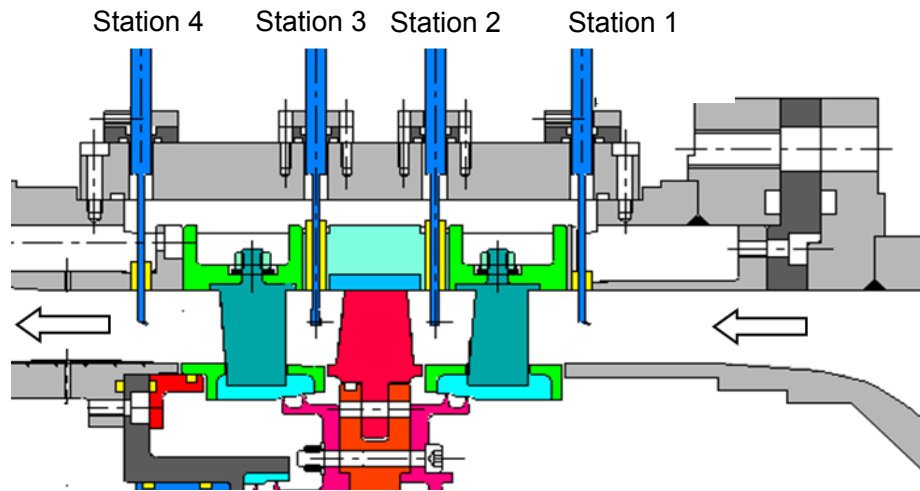


Figure 3.6.: The four axial positions of measurement station in the compressor stage

At each axial position, the instrumentation can be placed at four circumferential positions. This allows verifying the circumferential uniformity of the flow in the test section as well as minimizing the number of tests by carrying out simultaneous measurements. The circumferential positions correspond to 0° , 90° , 180° and 270° . The radial distribution of the various aerodynamic parameters measured are obtained thanks to carriage systems installed on the test section, Fig.3.7. These systems, supporting the probes, give the opportunity to move the probes along the blade height. The radial position of the probe is controlled via a motorization and an encoder placed on the shaft of the system. The encoder gives an accuracy on the radial position equal to $25\text{pulses}/0.05\text{mm}$.

The calculation of the pressure ratio and the efficiency of the compressor stage is based on complete measurement maps over the entire blade height and over two blade passages at the outlet of the stage. The carriage system has then to be coupled with a circumferential traverse mechanism. This system consists of a circumferential rotation of the stators instead of probes. This allows avoiding the presence of grooves in the tip casing in case of circumferential displacement of

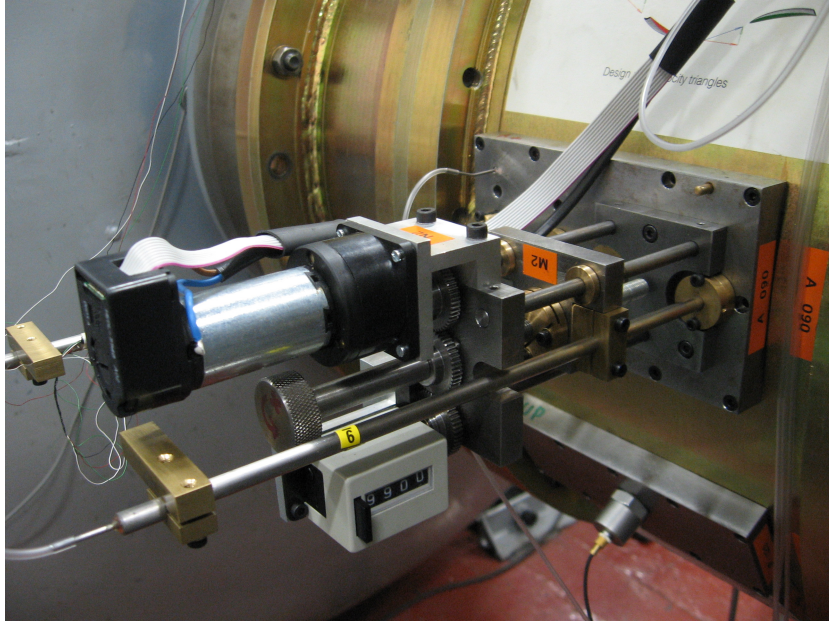


Figure 3.7.: Probe radial traversing system installed on the test section

probes, leading to high disturbances of the flow in the tip region. This mechanism is composed of three elements. The first one is a fastening system between the two stator rows, making them moving as a single block. The second element is the electrical motor coupled with a wheel-worm system allowing the circumferential displacement of the stators. And the last element is the encoder measuring the position of the stators with an accuracy of about 1% of the blade passage, or $0.15mm$. A schematic view of the displacement mechanism and the encoder is presented on Fig.3.8.

The test section is equipped with several wall static pressure taps. These are placed at various axial and circumferential locations as follows: 6 circumferentially distributed in the tip casing at the inlet of the stage, 6 in the hub casing at the inlet of the stage, 16 at the tip at the outlet of the stage, 6 and 16 holes are located two axial chords downstream the stator in, respectively, the tip and the hub casing, and finally 6 wall static pressure holes are positioned at the end of the divergent downstream of the test section. All these pressure holes allow the determination of the uniformity of the flow in the test section. This uniformity was studied by Edi Andrade in 2007 [3]. Furthermore, the wall static pressure holes, located at the inlet and the outlet of the stage, are used to complement the radial distribution of the total and static pressure at these two positions.

3.3. Aero-Dynamic Design of the First Stage of the Booster for Contra-Rotating Turbofan Architecture

51

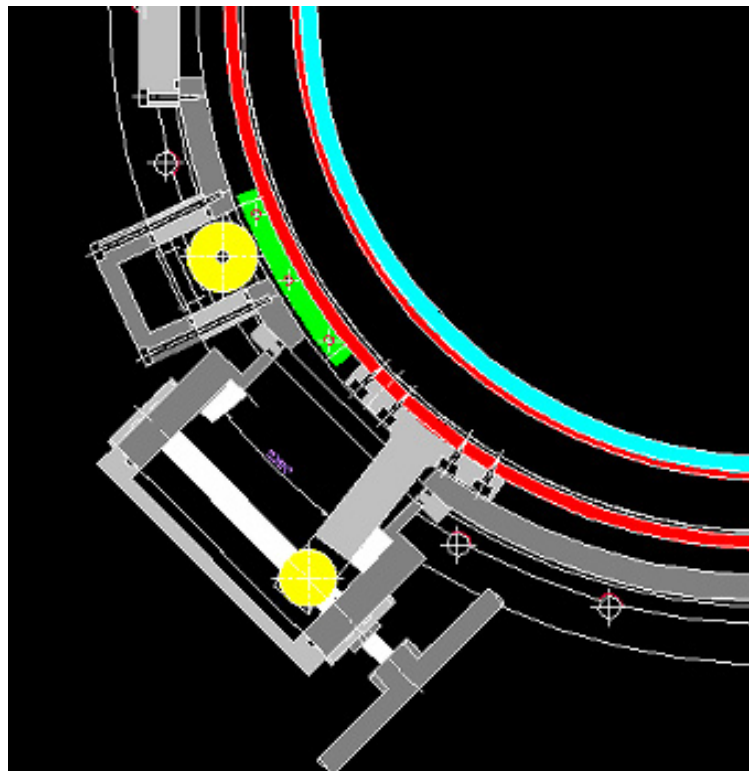


Figure 3.8.: schematic view of the circumferential displacement mechanism and its encoder

3.3. Aero-Dynamic Design of the First Stage of the Booster for Contra-Rotating Turbofan Architecture

As presented in chapter 1, the contra-rotating turbofan is characterized by a lower rotational speed of the low pressure components compared to classical engines. This leads to unusual high values of the loading- and flow coefficients. Furthermore, the higher mean radius of the booster in the case of contra-rotating architecture induces a lower aspect ratio of the booster blades. These elements have to be considered in the aerodynamic design of the compressor stage used for the experimental study.

The aerodynamic design of the compressor was done by Prof. J.-F. Brouckaert at the von Karman Institute according to the specifications from the industrial part-

ner Techspace Aero [13]. These specifications were obtained from the meridional design of the full scale booster, performed on the basis of a streamline curvature code including boundary layer calculation. Both rotor and stator solidity distributions were imposed by the specifications and conserved. The design of the blade profiles has been done at three different radii (10%, 50% and 90% of the blade height) by using an inviscid inverse design method starting from NACA profiles, modified to obtain controlled diffusion blades. Each blade profile performance was checked for design and off-design incidence by 2D Navier-Stokes calculations. To generate the three-dimensional blade shape, a simple radial stacking of the profiles has been adopted, using the center of gravity of all sections for the rotor as well as the leading edge for the IGV and stator, with a spline interpolation between sections. These calculations were performed with the turbomachinery code TRAF with a Baldwin-Lomax turbulence model [7].

In terms of similitude between the real booster and the model studied, the Reynolds number, the solidity of the rotor and the stator, the inlet Mach number, the non-dimensionalized mass flow as well as the non-dimensionalized rotational speed are conserved. The principal aerodynamic design characteristics are summarized in table 3.2.

Pressure Ratio (Cruise/Take-off)	1.14/1.15
Efficiency (Cruise/Take-off)	84/85 %
Power (Cruise/Take-off)	207/303 kW
Mass Flow (Cruise/Take-off)	9.03/22.68 kg/s
Hub to tip ratio (rotor inlet)	0.9
Average solidity (Rotor/Stator)	1.23/1.17
Aspect ratio (Rotor/Stator)	1.18/1.20
Reynolds number (Cruise/Take-off)	200 000/700 000
Rotor loading coefficient	1.04
Rotor Flow coefficient	1.58
Degree of Reaction	0.77

Table 3.2.: Stage characteristics of the VITAL test section

3.4. The Numerical Simulations Tools

The experimental results are compared to numerical simulations. This allows to assess the numerical capability to predict the global performance, the stability limit, the unsteady phenomena and the effect of an axisymmetric casing treatment in a low-pressure compressor. Furthermore, the possibility of the flow visualization allowed by the CFD gives a better understanding of the physical phenomena occurring in the compressor stage. Two CFD codes were used with various turbulence models. The comparison of the results of the turbulence models allows to highlight

the impact of the model on the results and to choose the most appropriate model for investigation of the effect of the casing treatment.

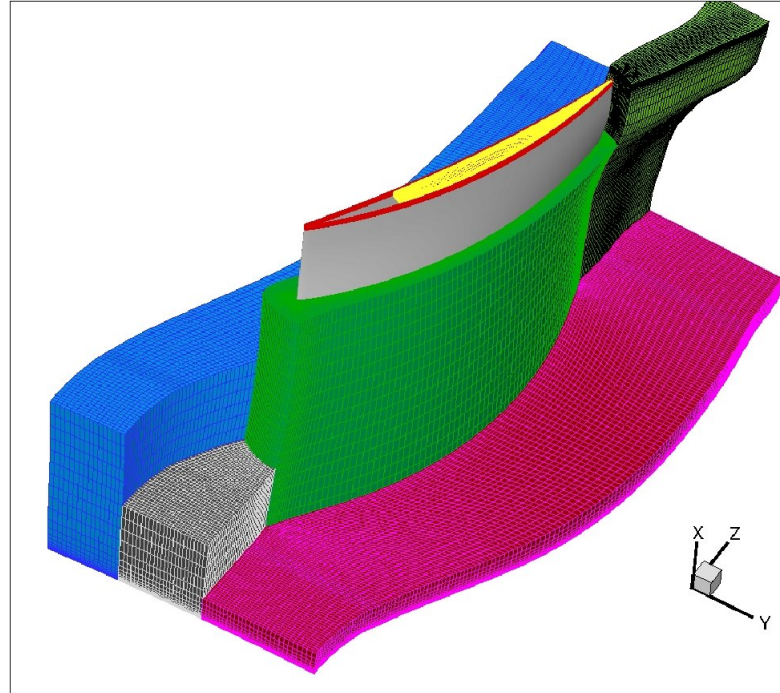


Figure 3.9.: Mesh topology around the rotor blade

The first numerical code is the Fine Turbo software developed by NUMECA International. This code solves the Reynolds Averaged Navier-Stokes equations using a cell centered finite volume method and implementing a 4-step Runge-Kutta method with an implicit (Zhu) residual smoothing phase for the (pseudo-)time discretization and a central Scheme with a Jameson type dissipation for the space discretization.

The meshes of each blade row have been generated with the AutoGrid5 software. Around the blades, they are based on a general O4H default topology with matched periodicity combined with a butterfly OH grid in the tip gap as shown on Fig.3.9. The mesh counts 0.85 million grid points for the IGv, 1.6 million for the rotor and 0.9 million for the stator. The global mesh holds about 3.35 million grid points. The average value of the dimensionless wall distance at the first nodes from the surfaces is around 1.

Regarding inlet conditions, the measured total pressure and total temperature profiles were imposed at the inlet of the compressor stage. As stage outlet bound-

ary condition, a radial equilibrium throttle condition was applied, fixing the static pressure at the shroud. For all the computations, the Yang-Shih $k - \epsilon$ turbulence model [117] was used and the measured turbulence level distribution was imposed at the inlet of the stage for all throttling conditions.

The second code was used by our partner CENAERO. This code is the elsA software (Ensemble Logiciel de Simulation en Aerodynamique) developed at ONERA. The software AutoGrid5 was used for the generation of the meshes of each blade row as well, and their topology is the same as for the FINE code. The global mesh holds about 4 million grid points, 1.1 million grid points for the IGV and the stator and 1.8 million for the rotor. In terms of boundary conditions, the same parameters were used as for the other code. Several turbulence models were used for the elsA code: the Launder-Sharma $k - \epsilon$ model [68], the Smith $k - l$ model [98] and the $k - \omega$ model of Wilcox [111].

The simulations performed with the two codes are fully turbulent. The four turbulence models used for the simulation of the compressor are two equations models. This means that these models include two transport equations for the representation of the turbulent properties of the flow. The $k - \epsilon$ model is based on the turbulent kinetic energy k , giving the energy in the turbulence, and the turbulent dissipation ϵ , representing the scale of the turbulence. This model was developed for finding an alternative to the algebraic models that are not suited for the simulation of moderate to high complexity flows. The equations of this model contain several constants. The values of these constants were determined empirically and are dependent on the developer of the model. This model is particularly efficient for free-shear layer flow with a relatively small pressure gradient. The accuracy of the predictions when using this model has been shown experimentally to be wanting for internal flows with large pressure gradients and flow separations. This makes this model inappropriate for the simulations of compressors.

The turbulence model $k - \omega$ is based on the turbulent frequency ω . The treatment of the near-wall region is improved by the switching from a wall function to a Low-Reynolds formulation. This model shows higher performance for wall-bounded configurations, transition, high adverse pressure gradient, and under low Reynolds conditions. Furthermore, this model has a higher numerical stability. Nevertheless, the flow separation is predicted to be excessive and early by this model.

The $k - l$ model is based on the characteristic length of the large scale of turbulences l . This term remains linear at the edge of the viscous layer whereas ϵ presents a very steep maximum [65]. Numerically, the regularity of l makes this model more favorable. Furthermore, this model has a low sensitivity to the distribution of the grid points close to the wall, and the compressibility is better taken into account by the model $k - l$ than by the $k - \epsilon$ model. Nevertheless, Castillon et al. [16] showed that the $k - \epsilon$ model is more appropriate for the simulation of transonic compressor but requires a higher CPU time computation.

A large investigation of the performance of various turbulence models in turbomachinery applications was presented by Dano in 2003 [23]. In this work, Dano

investigated the simulation of the tridimensional unsteady flow in turbine stages with various two equations turbulence models, including $k - \epsilon$, $k - l$ of Smith and $k - \omega$ of Wilcox. This investigation is based on the comparison of the numerical results with experimental measurements. The author has concluded that the model $k - l$ is particularly good for the simulation of the flow structures and phenomena present in turbomachines. At the same time, Dano showed that the model $k - \epsilon$ has a low robustness in these configurations, and that the model $k - \omega$ gives very poor results.

3.5. Test Conditions

The working conditions of the compressor were determined in order to keep the similarity between the real booster of a contra-rotating turbofan architecture and the reproduction at scale 1:2, according to the specifications given by Techspace Aero.

In terms of rotational speed, the similarity calculation gives a reduced rotational speed N_{Red} equal to 6.613. The reduced rotational speed is defined as:

$$N_{Red} = \frac{NDR}{\sqrt{\gamma RT_{01}}}, \quad (3.1)$$

where N is the mechanical rotational speed of the compressor, D is the diameter of the rotor, γ is the specific heat ratio, R the perfect gas constant and T_{01} the inlet flow temperature. The reduced rotational speed is then dependent on the inlet flow temperature. At 293K the rotational speed giving a reduced rotational speed of 6.613 is 4728RPM. But as the temperature is not constant, the mechanical rotational speed has to be adjusted continuously. This adjustment is nevertheless lower than 0.1% of the nominal speed.

The R4 closed loop facility allows the pressurization of the test section from 300mbars absolute to maximum 3bars absolute. This allows studying the effects of the Reynolds number on the compressor performance. In the case of the study of a compressor of an aircraft engine, as carried out in this thesis, two Reynold numbers are particularly interesting. They correspond to the cruise and the take-off regime. Under cruise conditions, the total pressure at the inlet of the compressor stage, giving the same Reynolds number as in the real engine, is equal to 1010mbar absolute, corresponding to nearly the atmospheric pressure. The Reynolds number corresponding to the take-off regime of an aero-engine is reached by the pressurization of the loop. The similarity calculation for the Reynolds number gives a pressure of 3.5bars absolute at the inlet of the compressor stage. But under these pressure conditions, the power absorbed by the compressor is equal to 303kW. On Fig.3.2, the maximum available power at 4728RPM is 241kW. The second Reynolds number studied was then obtained with a pressurization of the loop the 2.5bars instead of 3.5bars absolute as initially planned. The reduced rotational speed of the rotor is kept the same as for the cruise conditions.

The characterization of the compressor stage was performed at four operating points covering the entire operating range of the compressor. These points are defined as:

- Point A - Choke conditions. This point is reached with fully open throttle valve and corresponds to the maximum mass flow passing through the compressor.
- Point B - At this point, the pressure ratio from the fixed instrumentation located at mid-span is maximum.
- Point C - Design conditions. This point corresponds to the design parameters given by our industrial partner Techspace Aero.
- Point D - Near stall conditions. This point is the last stable point before the compressor runs into stall.

3.6. The Circumferential Groove Casing Treatment

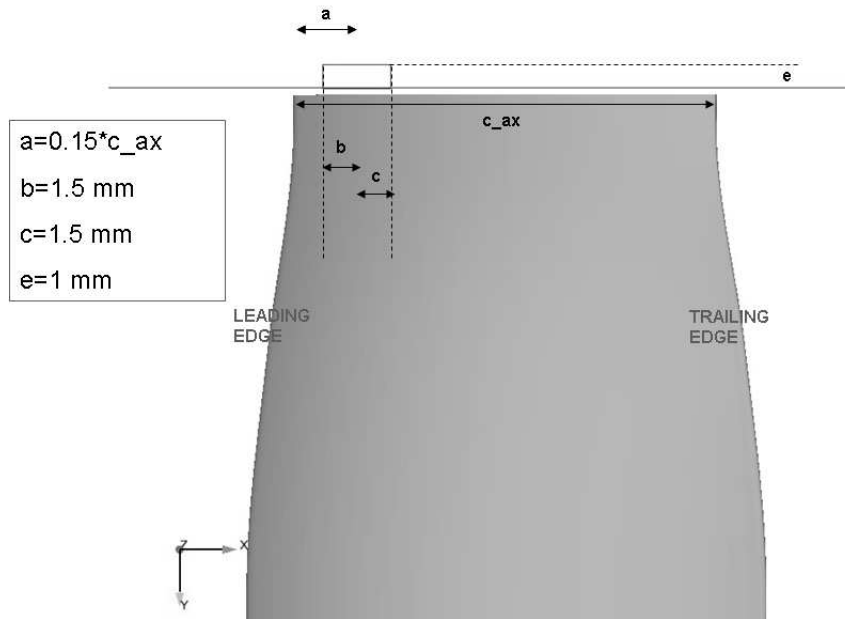
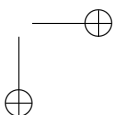
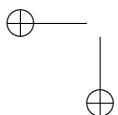
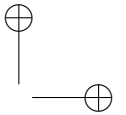
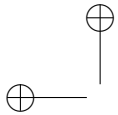


Figure 3.10.: Circumferential groove geometry

The circumferential groove casing treatment investigated in this study was chosen primarily on the basis of industrial considerations. Circumferential grooves are probably the simplest and most cost effective geometry to apply on a real engine as far as machining is concerned. Furthermore this kind of casing treatment allows an improvement of the stall margin with the smallest efficiency- or pressure ratio decrease, which is an essential requirement for engine applications. The stall margin improvement with a circumferential groove is usually lower than with other casing treatments but is generally sufficient for aircraft engine application. The greater mechanical integrity of grooved casing treatments is also an important aspect.

The geometry of the groove was also dictated by real engine considerations like forward axial displacement of the rotor, rub-in upon hard landing, erosion or fouling. The basic geometry selected by the industrial partner (Techspace Aero) is therefore somewhat away from the usual design recommendations that may be found in literature so far. Furthermore the choice of the exact geometry of the circumferential groove is based on a parametric study performed by our partner Cenaero. For this parametric study, the turbulence model used was Launder-Sharma $k - \epsilon$ and all the computations were performed at constant mass flow, corresponding to the last stable point of the smooth casing configuration. In all computations with the circumferential groove casing treatment, the structured groove mesh is connected to the main channel with the chimera technique. From the parametric study, a groove depth of 1 mm was chosen, which corresponds to 4.5% of the blade height or 4 to 8 times the tip gap, as presented here above. The length or axial extent of the groove was chosen to be 3mm (12.7% of rotor axial chord), i.e. with an aspect ratio of 1/3. The configuration retained is presented on Fig.3.10.



Chapter 4.

Overall Performance of the Compressor Stage

This chapter sets out the experimental overall performance of a single stage low pressure axial compressor designed for a counter-rotating turbofan engine architecture. The single stage is representative of the first stage of the booster of this particular engine architecture. As presented in the introduction of this thesis, this compressor stage is characterized by unusually high values of both loading- and flow coefficients.

The determination of the global performance of a compressor is based on steady measurements of pressure, temperature, velocity and flow direction. First comes a description of the specific instrumentation used for these measurements, including the calculation of the uncertainties on the measurements. In addition, there is a presentation of the hot-wire measurements used for the flow turbulence characterization.

The flow-conditions measured at the inlet of the compressor stage were imposed as inlet conditions in the numerical simulations. These conditions, including the boundary layer size, the turbulence level and the radial distribution of the flow characteristics, are detailed in this chapter.

The experimental performance of the compressor stage in terms of mass flow, pressure ratio, efficiency and stability margin is presented and compared to the results from the numerical simulations for two Reynolds numbers, corresponding to the cruise and the take-off conditions. The comparison of the experimental results with the numerical simulations, performed with various turbulence models, allows determining the model matching best the measurements. These characteristics of the compressor are derived from the mapping of the flow parameters at the inlet and the outlet of the stage.

A description of the performance of the rotor alone is given and also compared to the numerical simulations. The determination of the Greitzer criterion for this particular compressor will give us preliminary information about the instability mode that occurs in the compressor.

The influence of the rotor casing configuration on the overall performance of the compressor stage is studied. The two casing architectures considered are the smooth casing, representative of the initial configuration, and a circumferential groove casing treatment.

4.1. Instrumentation for the Overall Performance Determination

The very small size of the flow passage in the test section (blade height of about $23mm$) led to the manufacturing of specially designed probes with careful attention to the miniaturization. For the same reason, the use of the classical pressure and temperature rakes, both at the inlet and the outlet of the stage, has been avoided. In consequence, the completion of full area probe traverses were required, in order to establish the performance data of the compressor.

4.1.1. Steady Pressure Measurements

All the steady pressure measurements were performed using the variable reluctance relative pressure transducers from the Validyne Engineering company. The principle of this type of transducer is based on the variation of a magnetic field due to the displacement of a metallic diaphragm. The diaphragm is positioned between two magnetic cores wound with induction coils [4]. A pressure difference between the two faces of the diaphragm bends it, inducing a variation of the reluctance of the magnetic field and thus a change of the induction in the coils. These transducers present a high robustness, a good stability and a large measurement range. Because of the presence of the cavities on both sides of the diaphragm, the frequency response of this type of transducer is very limited and its use is restricted to steady pressure measurements. As to the reference pressures, the atmospheric pressure is taken into account for the tank pressure measurement and the tank pressure for all the others. This system of reference pressures allows decreasing the pressure difference between the two sides of the sensors, leading to the use of thin membranes, characterized by a higher accuracy on the measurement.

The in-situ static calibration of the pressure transducer is performed at the beginning of each measurement day and is based on the pressurization capability of the facility. These calibration parameters are determined by means of the acquisition of the signals from the various transducers at two different pressure levels. During the static calibration, the reference of all the transducers were connected to the atmospheric pressure. The first pressure level is the atmospheric pressure reached by opening the facility to the atmosphere, *i.e.* the pressure difference between the reference and the measurement point is equal to zero for all the transducers. The pressure of the second calibration point is equal to $100mbar$. This pressure level is reached by closing and pressurizing the facility, the reference pressure of the transducers being kept at the atmosphere. The value of the pressure for the second calibration point is read on a mercury manometer. This procedure allows establishing the calibration curves for all the pressure transducers simultaneously. The comparison of the calibration parameters of the same sensors, determined on different days, shows that the offset and the slope vary respectively by less than $0.7mbar$ and less than 1.2%. The maximum variation corresponds to

long periods of inactivity. These values correspond to about five times the uncertainty on the reading of the pressure level on the mercury manometer. Indeed, this manometer being graduated with an accuracy of 0.1 mmHg , the reading error on the pressure value is equal to 0.1332 mbar . The variation of the slope and the offset is due to the electronic components of the measurement chain. This gives an idea about long term stability and shows the importance of a daily calibration of the transducers.

The steady static- and total pressures are measured at various locations in the test section. Fixed probes are used for the measurement of the inlet total pressure at mid-span as reference pressure. The wall static pressure is measured via pressure taps, and the radial distribution of the total pressure at various axial positions is established using moving pressure probes. All these pressure measurements allow the complete determination of the pressure distribution in the test section.

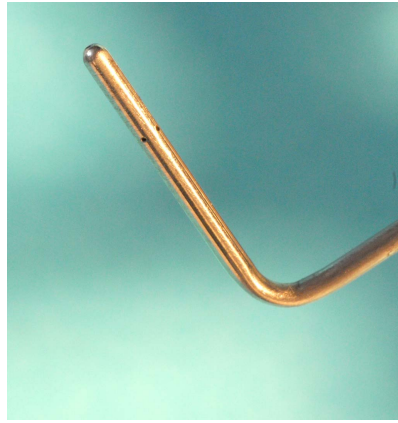


Figure 4.1.: The head of the Prandtl probe

The inlet total pressure measured at mid span is used as a reference for the pressure measurements. This pressure is measured using a fixed Prandtl probe, see Fig.4.1. This probe consists of a tube directed parallel to the airflow, with a hole at its tip and four holes distributed on the probe's circumference. The hole at the tip of the probe is used to measure the total pressure. The holes on the circumference of the probe are connected to a single port, and are used to measure static pressure, averaged over the circumferential holes.

The wall static pressure conditions at tip and hub were measured by the use of static pressure holes. Six holes at tip and hub located at the same axial position as the stage inlet measurement plane are distributed along the circumference of the test section. The holes of each radial position were connected together in order to get an averaged value of the static pressure over the circumference.

The radial distribution of the total pressure at the various axial positions is based

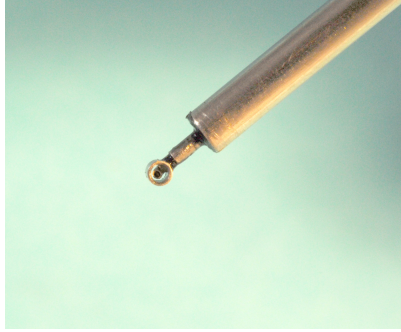


Figure 4.2.: The head of the Kiel probe

on a Kiel probe, Fig.4.2. This probe is a variation of the Pitot probe where the inlet tube is protected by a shroud. This arrangement allows a large reduction of the probe sensitivity to the flow angle. The exact sensitivity of the probe measurement to the flow angle was determined at various Mach numbers in the C4 free-jet facility at the von Karman Institute. Figure 4.3 gives the pressure ratio $P_{probe}/P_{reference}$ of a total pressure probe, where P_{probe} is the total pressure measured by the probe in the jet and $P_{reference}$ is the real total pressure of the jet measured in the settling chamber. This figure shows that the Kiel probes are insensitive within 1% to the flow angle in a range of $\pm 22.5^\circ$. An offset between the geometrical zero and the aerodynamic zero of 5° is visible. This comes from the manufacturing of the probe. On account of the miniaturization of the instrumentation, the inlet tube of the probe is not perfectly centered in the shroud, and the shroud is not exactly aligned with the inlet tube. During the measurement campaigns, the probes were always aligned to the mean flow direction taken from the compressor design values. This allows to stay in the angle insensitivity range of the probes, even neglecting the angular offset of 5° .

The Kiel probes, used in this project, have a very small size. The stem, the shroud and the support between them have a diameter of, respectively, 6, 1.5 and 1mm. This very high miniaturization of the instrumentation is imposed by the small height of the blades in order to avoid important blockage. Despite of this miniaturization, the size of the probes do not allow to perform measurement close the wall. In the case of the Kiel probes, the measurement are limited between 4 and 96% of the blade height.

The accuracy of the pressure measurement is depending on several elements because of the large number of components of the measurement chain. The absolute pressure P_i , at radial location i , is the sum of the pressure $P_{probe,i}$ measured at this point relative to the tank (since the pressure of the tank is taken as reference for all the pressure sensors), of the pressure in the tank measured relative to the

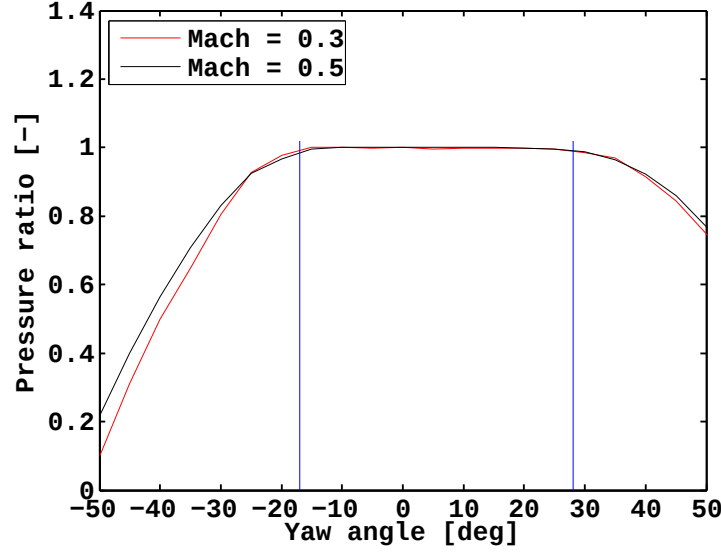


Figure 4.3.: Angular sensitivity of the total pressure probes

atmosphere P_{tank} , and the atmospheric pressure P_{atm} :

$$P_i = P_{probe,i} + P_{tank} + P_{atm} \quad (4.1)$$

From this equation, the uncertainty on the pressure measurement is computed from the following equation:

$$\delta P_i = \sqrt{(\delta P_{probe,i})^2 + (\delta P_{tank})^2 + (\delta P_{atm})^2} \quad (4.2)$$

The pressure of the probe is calculated by the linear relation between the voltage and the pressure determined by the static calibration $P_{probe} = Sl.V + O$, where S and O are respectively the slope and the offset of the calibration curve. The uncertainty on the pressure P_{probe} is given by:

$$\delta P_{probe} = \sqrt{(V.\delta Sl)^2 + (Sl.\delta V)^2 + (\delta O)^2} \quad (4.3)$$

The calculation of the uncertainty on the slope δS is based on the expression of the pressure sensitivity of the transducer $Sl = \frac{\Delta P}{\Delta V}$. The derivation of this

expression gives the uncertainty:

$$\delta Sl = \sqrt{\left(\frac{1}{\Delta V} \cdot \delta(\Delta P)\right)^2 + \left(\frac{-\Delta P}{\Delta V^2} \cdot \delta(\Delta V)\right)^2} \quad (4.4)$$

The values of the uncertainty on the pressure variation $\delta(\Delta P)$ and on the offset δO come from the static calibration of the probe and are depending on the uncertainty on the pressure level of the two calibration points δP_{calib} :

$$\delta(\Delta P) = 2 \cdot \delta P_{calib} \quad \text{and} \quad \delta O = \delta P_{calib}. \quad (4.5)$$

The uncertainty δP_{calib} comes from the reading of the pressure level on a mercury manometer. As explained above, this uncertainty is equal to 0.1 mmHg , corresponding to 0.1332 mbar . The atmospheric pressure P_{atm} used in equation 4.1 being measured on the same device, its uncertainty will be equivalent.

$$\delta P_{calib} = \delta P_{atm} = 0.1 \text{ mmHg} = 0.1332 \text{ mbar} \quad (4.6)$$

The uncertainty on the voltage variation $\delta(\Delta V)$ in equation 4.4 is equal to two times the error on the voltage due to the accuracy of the acquisition system δV . The acquisition system used for the measurement in this project is a 16 bits over a range of $\pm 10 \text{ Volts}$ giving an uncertainty on the voltage of:

$$\delta V = \frac{20V}{2^{16}} = 0.30518 \text{ mV} \quad (4.7)$$

The pressure in the tank relative to the atmospheric pressure is calculated the same way as for the pressure of the probe P_{probe} .

This procedure of the determination of the pressure uncertainty is applicable to all the pressure measurements. Except, of course, for the pressure in the tank where equation 4.1 is simplified by the removal of the term $P_{probe,i}$.

Finally, this calculation gives a value for the uncertainty of about 0.04% . Under Cruise conditions, the measured total pressures are between 1010 and 1210 mbar . The uncertainty on these values under cruise conditions are lower than 0.5 mbar .

4.1.2. Temperature Measurements

The total temperature measurements in the test section were performed using thermocouple probes presented on Fig.4.4. These probes have the advantages of being very small, inexpensive and presenting a good accuracy on the temperature measurement. The thermocouple principle is based on the "Seebeck" effect. This effect is the fact that in a metal a temperature gradient always goes together with a difference of potential. The difference of potential is depending on the temperature gradient and on the metal type. This allows to measure the "Seebeck" effect by means of a system composed by two different metals. The principle of the

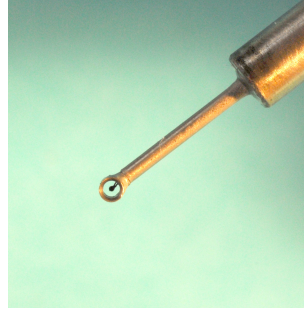


Figure 4.4.: The head of the shielded thermocouple probe

thermocouple measurement device is shown on Fig.4.5. The voltage between the two metallic wires is depending of the temperature difference between the junction (1) and the extremities of the wires (2) and (3). In case of direct measurement of the voltage, the extremities of the wires are connected to a voltage measurement device. The voltage measured is proportional to the temperature difference between the junction and the voltmeter. In order to avoid the influence of the temperature of the voltmeter, a reference temperature area, characterized by a constant temperature, is included in the circuit. The extremities of the two wires are connected to identical material wires and these two new junctions are placed in the reference temperature area. The voltage is then measured at the extremities of these two new wires. As these two wires are composed of the same material and their extremities are at the same temperature (reference and voltmeter), their influence on the global voltage difference is null. This difference is thus only depending on the temperature difference between the reference temperature (2 and 3) and the junction temperature (1), [92].

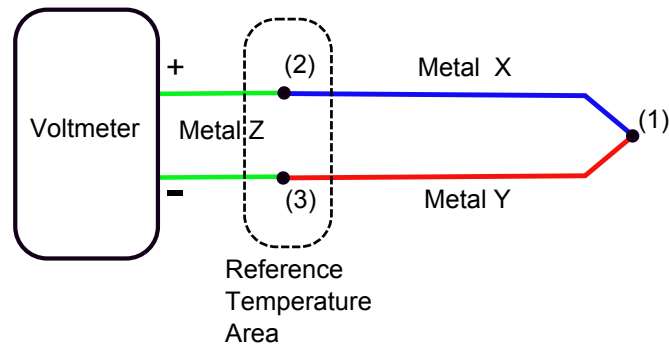


Figure 4.5.: Thermocouple measurement circuit with reference temperature area

In the frame of this work, the thermocouple probes used are Chromel-Alumel thermocouples (type K) with a junction diameter of about $100\mu m$. This type of thermocouple has a temperature range of $1520K$ with a sensitivity of $40\mu V$ per Kelvin. The very low level of the voltage imposes the use of an amplification system in the measuring chain to get a sensitivity of $100mV$ per Kelvin. The reference temperature area is an integrated circuit called cold junction compensator with an artificial temperature of $273.15K$. The dimension of the thermocouple probes are similar to the Kiel probes, leading to a similar measurement radial range.

The principle of the thermocouple is based on the determination of the temperature of an environment via the measurement of the temperature of the wires' junction. In the case of gas applications, the temperature of the junction is different from the temperature of the gas and depends on the heat transfer mechanism between the environment and the thermocouple probe:

- heat transfer between the walls and the probe by radiation,
- heat transfer by conduction along the wires and the probe,
- conversion of kinetic energy into thermal energy in the boundary layer around the thermocouple.
- heat transfer from the boundary layer to the junction by convection

Three types of measurement error result from these heat transfer mechanisms, [60]. The conduction error and the radiation error come from the conduction and radiation phenomena. The last source of error is the velocity of the flow and is called velocity error. Ideally, the flow total temperature should be obtained by bringing the flow to rest adiabatically, allowing the conversion of the kinetic energy of the fluid in thermal energy. In reality, the gas is not stopped and its deceleration by the presence of the probe is not adiabatic. A significant portion of the kinetic energy is lost through viscous dissipation, whose amount is dependent on the geometry of the immersed body and the molecular Prandtl number of the fluid. This causes the velocity error.

In order to take these velocities into account in the determination of the gas temperature, the recovery factor is defined as:

$$r = \frac{T_J - T_S}{T_0 - T_S} \quad (4.8)$$

with T_0 the total temperature of the gas, T_J the temperature of the thermocouple junction and T_S the static temperature of the flow. This equation can be written in function of the Mach number by using the relation of the temperatures for adiabatic and perfect gases [101]:

$$r = 1 - \left(1 - \frac{T_J}{T_0}\right) \cdot \frac{2 + (\gamma - 1) M^2}{(\gamma - 1) M^2} \quad (4.9)$$

where M is the Mach number and γ is the specific heat ratio. This ratio is equal to 1.4 for dry air at 293K. A well designed thermocouple probe has a recovery factor close to 1. The use of a shield around the thermocouple junction allows the reduction of the radiation and velocity errors. This makes the recovery factor to increase.

In accordance with these relations, the total temperature of a flow measured by a thermocouple in function of the junction temperature is given by the equation:

$$T_0 = \frac{T_J}{\left[1 - (1 - r) \cdot \frac{(\gamma - 1)M^2}{2 + (\gamma - 1)M^2}\right]} \quad (4.10)$$

Subsequently to the principle of the thermocouple probes, the calibration of these probes has to be performed in two steps. First, the relation between the temperature of the junction of the thermocouple and the voltage measured at the outlet of the amplifier has to be established. This is the static calibration. This part is done by means of an oil bath with a mercury thermometer as reference. During the various test campaigns performed in the frame of this project, four thermocouple probes have been used. Two fixed probes were placed at mid-span at the stage inlet (Tprobe5) and at the stage outlet (Tprobe6) as reference temperatures. The two others were used for the determination of the radial distributions of the total temperature at various axial positions (Tprobe7 and Tprobe8). These probes were designed in order to be placed on the traversing carriages. Figure 4.6 shows the calibration curves of the thermocouples used for this project.

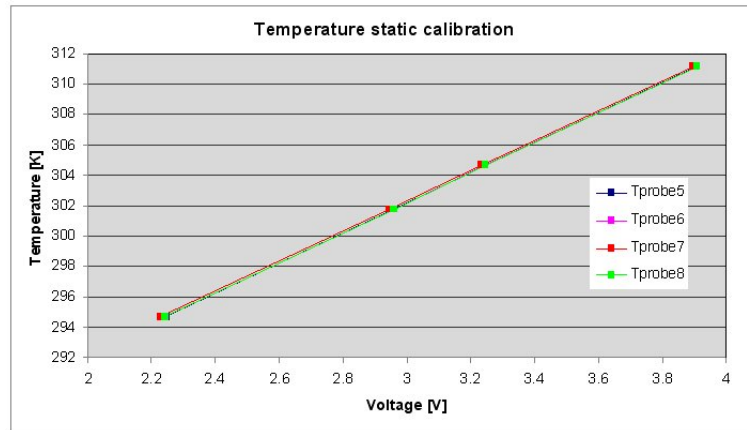


Figure 4.6.: Static calibration curves for the various thermocouples used in the VITAL project

The second part of the calibration allows the determination of the recovery factor in order to take into account the influence of the velocity errors on the

temperature measurement. This calibration was performed in the C4 calibration facility at the von Karman Institute. The recovery factor determination is based on equation 4.9. Figure 4.7 shows the recovery factor for the four thermocouple probes.

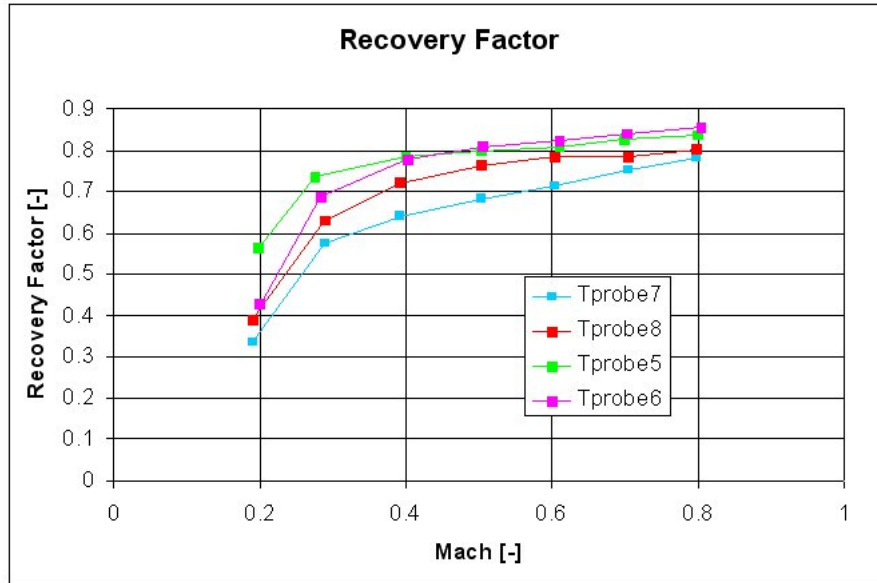


Figure 4.7.: Recovery factor as a function of the Mach number for the various thermocouples used in the VITAL project

A very high dispersion of the calibration curves of the recovery factor is visible on this figure. Two elements can explain this phenomenon. The first element is the uncertainty on the recovery factor. As presented below, the uncertainty on the recovery factor is dependent on the Mach number. At low Mach number, the uncertainty is generally higher than the value of the recovery factor. A part of the dispersion at low Mach number is then ascribable to the uncertainty on the recovery factor.

The second element explaining the high dispersion is the dependence of the recovery factor of a thermocouple on its design. This was studied by Paniagua [86] in 2002. The ratio between the length and the diameter of the wires as well as the ventilation ratio of the shield have an influence on the recovery factor. Furthermore, the size of the junction has a strong importance in the gas-junction convection phenomenon. The thermocouple probes used in the frame of this work were manually manufactured at the von Karman Institute. Variations of the three geometrical parameters cited above exist and can be important. These variations

lead to a high variation of the recovery factor of the various probes.

From equation 4.10, and assuming a constant adiabatic ratio γ without error, the uncertainty on the total temperature measurement by a thermocouple probe in the test section is a function of the uncertainty on the temperature of the thermocouple junction, on the recovery factor and on the Mach number of the flow. This uncertainty is given by the relation:

$$\delta T_0 = \sqrt{\left(\frac{\partial T_0}{\partial T_J} \cdot \delta T_J\right)^2 + \left(\frac{\partial T_0}{\partial r} \cdot \delta r\right)^2 + \left(\frac{\partial T_0}{\partial M} \cdot \delta M\right)^2}. \quad (4.11)$$

With

$$\frac{\partial T_0}{\partial T_J} = \frac{1}{1 - (1 - r) \cdot \frac{(\gamma - 1) \cdot M^2}{2 + (\gamma - 1) \cdot M^2}}, \quad (4.12)$$

$$\frac{\partial T_0}{\partial r} = -\frac{T_J \cdot \frac{(\gamma - 1) \cdot M^2}{2 + (\gamma - 1) \cdot M^2}}{\left(1 - (1 - r) \cdot \frac{(\gamma - 1) \cdot M^2}{2 + (\gamma - 1) \cdot M^2}\right)^2} = -\frac{T_J \cdot (\gamma - 1) \cdot M^2 \cdot (2 + (\gamma - 1) \cdot M^2)}{(2 + r \cdot (\gamma - 1) \cdot M^2)^2} \quad (4.13)$$

$$\frac{\partial T_0}{\partial M} = \frac{T_J \cdot (1 - r) \cdot 4 \cdot (\gamma - 1) \cdot M}{(2 + r \cdot (\gamma - 1) \cdot M^2)^2}, \quad (4.14)$$

The development of the last equation is given in the App.A.

The uncertainty on the junction temperature measurement δT_J is determined by the linear relation between the voltage measured by the acquisition system and the temperature, established by the static calibration:

$$T_J = Sl \cdot V + O \quad (4.15)$$

In accordance with this equation, the determination of δT_J follows the same procedure as for the uncertainty on pressure presented in the previous section. The only parameter that is changed is the uncertainty on the reference level during the static calibration. In the case of the temperature, this uncertainty comes from the reading of the reference temperature on a mercury thermometer graduated to $0.1K$, so it is assumed that:

$$\delta T_{calib} = 0.05K. \quad (4.16)$$

The uncertainty on the recovery factor δr , in equation 4.11, has two origins. Indeed the recovery factor is a function of the Mach number $r = f(M)$. The uncertainty on r is due to the uncertainty on the Mach number at the measurement point δM on the one hand, and to the uncertainty on the function $f(M)$ from the calibration, $\delta f(M)$, which is also a function of M , on the other hand. The global uncertainty on the recovery factor is then the maximum of the absolute value of the difference between the value of the recovery factor r and the value of this factor plus or minus the uncertainty on function $\delta f(M)$ over the Mach number error range, as presented on Fig.4.8:

$$\delta r = \max \left(\text{abs} [f(M) \pm \delta f(M)]_{M - \delta M}^{M + \delta M} \right) \quad (4.17)$$

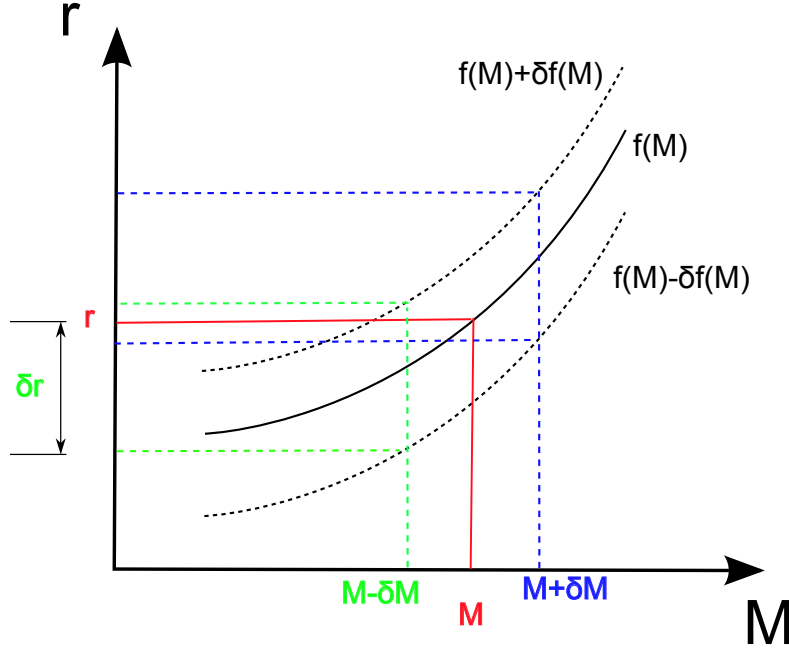


Figure 4.8.: Uncertainty on the recovery factor due to the uncertainty on the Mach number

From equation 4.9, the uncertainty on function $f(M)$, giving the recovery factor in function of the Mach number, is caused by the uncertainty on the reference total temperature $T_{0,cal}$, the uncertainty on the junction temperature $T_{j,cal}$ and the uncertainty on the Mach number M_{cal} during the calibration at the C4 VKI facility:

$$\delta r = \sqrt{\left(\frac{\partial r}{\partial T_0} \cdot \delta T_0\right)^2 + \left(\frac{\partial r}{\partial T_j} \cdot \delta T_j\right)^2 + \left(\frac{\partial r}{\partial M} \cdot \delta M\right)^2}. \quad (4.18)$$

This equation can be solved using the following relations:

$$\frac{\partial r}{\partial T_0} = \frac{-\frac{T_j}{T_0^2}}{\left(\frac{(\gamma-1) \cdot M^2}{2 + (\gamma-1) \cdot M^2}\right)}, \quad (4.19)$$

$$\frac{\partial r}{\partial T_j} = \frac{1}{\left(T_0 \cdot \frac{(\gamma-1) \cdot M^2}{2 + (\gamma-1) \cdot M^2}\right)}, \quad (4.20)$$

$$\frac{\partial r}{\partial M} = \left(1 - \frac{T_j}{T_0}\right) \cdot \left(\frac{4}{(\gamma-1) \cdot M^3}\right). \quad (4.21)$$

The uncertainties on the reference and junction temperatures come from the static calibration of the two thermocouple probes. This uncertainty has been presented earlier. The uncertainty on the Mach number in the C4 facility comes from the pressure uncertainties. Indeed, the Mach number in this facility is determined by means of the relation between the Mach number and the static-to-total pressure ratio of an adiabatic perfect gas [101]:

$$M = \sqrt{\frac{2}{\gamma - 1} \cdot \left[\left(\frac{P_0}{P_s} \right)^{\frac{\gamma - 1}{\gamma}} - 1 \right]}, \quad (4.22)$$

In this relation, the total pressure P_0 of the jet is measured in the settling chamber located upstream the convergent nozzle of the facility, and the atmospheric pressure is considered as static pressure P_s , since it is an open jet. The differential derivation of this equation gives the uncertainty on the Mach number in the C4 facility:

$$\delta M = \sqrt{\left(\frac{1}{M \cdot \gamma \cdot P_s \cdot \left(\frac{P_0}{P_s} \right)^{\frac{1}{\gamma}}} \cdot \delta P_0 \right)^2 + \left(-\frac{P_0}{M \cdot \gamma \cdot P_s^2 \cdot \left(\frac{P_0}{P_s} \right)^{\frac{1}{\gamma}}} \cdot \delta P_s \right)^2}. \quad (4.23)$$

In equation 4.11, the last term refers also to the Mach number. But it is not the same Mach number. In this case, the Mach number is the measured one at the temperature measurement point in the compressor stage. This measurement is performed by means of a three-hole probe. This probe and the uncertainty on the Mach number are presented in the next section.

The determination of the uncertainty on the temperature measurement, following the procedure described above, give a value around $0.3K$. This high level of uncertainty is caused principally by the correction of the temperature with the recovery factor and the very large uncertainty on this factor.

4.1.3. Flow Direction, Mach Number and Static Pressure Measurements

The flow direction α , Mach number M as well as the static pressure P_s were measured by means of a miniature three-hole probe. This probe is constituted by three pipes placed side by side in the same plane, see Fig.4.9. These pipes are cut following different directions. The central pipe, called P_c , is cut perpendicularly to its axis, and the two others, called P_L and P_R , are cut, respectively, at $\pm 45^\circ$ as shown on Fig.4.10. These tubes have a $0.6mm$ external diameter and the total width of the head is $2mm$. The radial range of measurement with this probes is from 1 to 96% of the blade height.

The determination of the flow direction, flow velocity and the static pressure is based on the definition of four coefficients expressing the sensitivity of the probe to these parameters [11]. The coefficients give a relation between the three pressures



Figure 4.9.: The miniature three-hole probe used for the determination of the radial distributions of the velocity, flow angle and static pressure

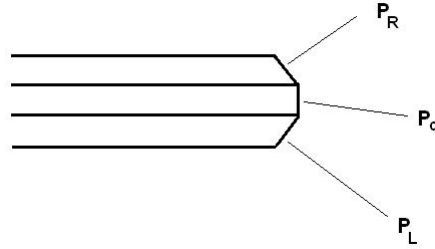


Figure 4.10.: Three-holes probe schematic view

P_c , P_L and P_R from the probe on the one hand, and the flow angle α , the Mach number M , and the total and static pressure P_0 and P_s on the other hand:

- Angular Sensitivity

$$K_{yaw}(\alpha, M) = \frac{P_L - P_R}{P_c - \frac{P_L + P_R}{2}} \quad (4.24)$$

- Pressure Sensitivity

$$K_{Tot}(\alpha, M) = \frac{P_t - P_c}{P_c - \frac{P_L + P_R}{2}} \quad (4.25)$$

$$K_{Dyn}(\alpha, M) = \frac{P_t - P_s}{P_c - \frac{P_L + P_R}{2}} \quad (4.26)$$

- Mach number Sensitivity

$$K_{Mach}(\alpha, M) = \frac{P_c}{P_c - \frac{P_L + P_R}{2}} \quad (4.27)$$

The values of these coefficients are only dependent on the geometry of the probe. They were determined experimentally during a calibration phase in function of the flow angle α and the Mach number M . This calibration was performed in the C4 calibration tunnel of the von Karman Institute over an angular range from -20° to 20° in steps of 2° , and over a Mach number range from 0.2 to 0.8 in steps of 0.1. The coefficient maps were refined by means of a linear interpolation method. Finally this method allows the determination of the four coefficients over the same range of angles and Mach numbers, and with a refinement of 0.5° and 0.01 for the Mach number. The interpolated values of the four coefficients in function of the Mach number M and the flow angle α are presented on Fig.4.11.

The determination of the flow angle, flow velocity and static pressure in the compressor stage is then based on the three pressures from the three-hole probe P_c , P_L and P_R , and on the coefficient maps from the calibration. Given the independence of the four coefficients, the determination procedure is iterative.

Firstly, the values of the coefficients $K_{yaw,m}$ and $K_{Mach,m}$, at the measurement point m , are calculated from the pressures measured, using equations 4.24 and 4.27. Considering the calculated $K_{yaw,m}$ and an initial Mach number M_1 of 0.5, a first angle α_1 is determined by means of the calibration map of coefficient K_{yaw} . Using this angle α_1 , the value of coefficient $K_{Mach,m}$ and the calibration map of coefficient K_{Mach} , a new Mach number M_2 is determined. This procedure is then repeated with the new value of the Mach number until the difference between the new Mach number M_i and the previous one M_{i-1} is inferior to 0.01. The flow angle and Mach number are then equal to the last calculated α_i and M_i . This iterative method is really fast, usually less than 4 iterations are required to reach the criterion of convergence. This is due to the quasi independence of coefficient K_{yaw} from the Mach number. The two calculated values of the Mach number and flow angle are then used in order to determine the coefficients $K_{Tot,m}$ and $K_{Dyn,m}$ from the calibration maps of these two coefficients. Finally, the static and total pressures at the measurement point are calculated by means of the equations 4.25 and 4.26 and with the calculated values of K_{Tot} , K_{Dyn} , and the measured values of P_c , P_L and P_R .

The effect of the pitch angle on the three-holes probe has been completely neglected. This comes from the very low pitch angle in the compressor stage, lower than $\pm 5^\circ$ according to the CFD computation, and the negligible difference in the calibration curves performed with a pitch angle of 0° , 5° and -5° .

4.1.4. Three-hole Probe Measurements Uncertainty

The determination of the uncertainty on the angle, Mach number, total and static pressure measured with the three-holes probes is complicated and required some simplifications. The first approximation consists to suppose the angle proportional to K_{yaw} and independent from the Mach number. Then, the uncertainty on the Mach number can be determined from the maximum variation due to the uncertainty on the angle and on the calibration. The uncertainties on the pressures

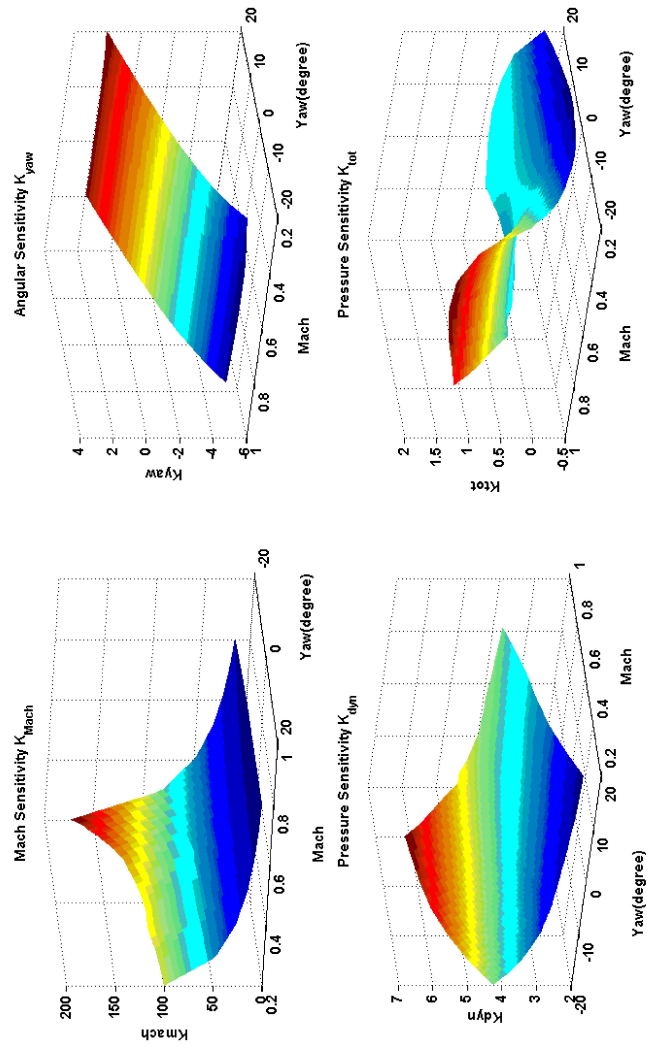


Figure 4.11.: Calibration maps of the three-hole probe sensitivity coefficients as a function of Mach number and flow angle

are derived by using the classical partial derivation method. The procedure of the calculation of the uncertainty on the measurement of a three-holes probe is described in details in Appendix A.

This procedure gives an uncertainty on the angle measurement under cruise conditions of about 0.2° , on the Mach number of 1.5%, and on the static and total pressure under cruise conditions of about 0.4mbar .

4.1.5. Hot Wires

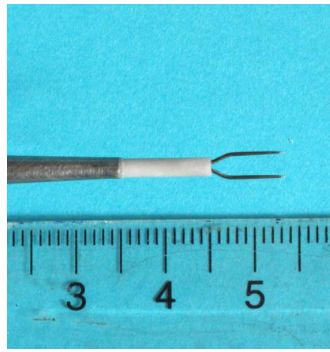


Figure 4.12.: The head of the hot-wire probe

The hot-wire probe (Fig.4.12) or hot-wire anemometer is one of the oldest measuring systems used in experimental fluid dynamics. One of the main advantages of the hot-wire anemometer is its very high frequency range. The hot-wire probes were extensively studied by Bruun [14]. The hot-wire probe consists of two main parts. The first is a short (1 to 3 mm) conductive wire ($5\mu\text{m}$ in diameter), generally a platinum coated tungsten wire, placed in the flow. The second part of the anemometer is composed of the electronic equipment. This equipment is used for the power supply of the wire, and for the conversion of the wire output into a usable signal. A current, supplied by the electronics, passes through the wire. This current allows to heat the wire up to a fixed temperature. This temperature is maintained constant by varying the voltage difference between the two extremities of the wire, which entails a variation of the current. The heat transfer from the hot-wire to the flow increases with increasing flow velocity. At constant flow temperature, the flow velocity has a direct influence on the wire temperature, meaning that a relation can be established between the flow velocity and voltage over the wire.

For constant temperature anemometers, due to the thermal equilibrium between the flow and the wire, the heat transfer from the hot-wire to the air flow is equal to the heat generated by the current flowing through the wire. The steady state

energy balance is expressed by:

$$Q_{in} = I_w^2 R_w = \pi d l h (T - T_f) \quad (4.28)$$

where Q_{in} is the heat dissipated in the wire, I_w is the current going through the wire, R_w is the resistance of the wire at temperature T , d and l are respectively the wire diameter and length, T_f is the temperature of the fluid and h is the heat transfer coefficient. By using the definition of the Nusselt number Nu in function of h and k , the thermal conductivity coefficient of the fluid:

$$Nu = \frac{hD}{k} \quad (4.29)$$

and the voltage over the wire E_w , equation 4.28 can be rewritten as:

$$I_w^2 R_w = \frac{E_w^2}{R_w} = \pi l k (T - T_f) Nu \quad (4.30)$$

This relation shows the link between the Nusselt number and the current passing through the wire. This Nusselt number is depending on various factors: the fluid velocity, the temperature difference between the wire and the fluid, the physical properties of the fluid and the probe geometry. A dimensional analysis gives a relation between the Nusselt number, the Reynolds number and the Prandtl number :

$$Nu = f(Re, Pr) \quad (4.31)$$

where $Re = \text{Reynolds number} = \frac{UD}{\nu} = \frac{UD\rho}{\mu}$

and $Pr = \text{Prandtl number} = \frac{C_p \mu}{k}$

In accordance with the definition of the Reynolds number and the Prandtl number, and to equation 4.31, relation 4.30 shows that the current is a function of the flow velocity and flow density.

Assuming constant resistance of the wire (by constant wire temperature), flow temperature and flow density, equation 4.30 can be written as a relation between the voltage over the wire and the flow velocity through King's law:

$$E^2 = A + B.v^n \quad (4.32)$$

where A , B and n are constants obtained by calibration. This calibration was performed in the R4 VKI facility. The hot-wire probe was placed at mid-span at the inlet of the compressor stage. The compressor was then run at various rotational speeds in order to submit the probe to flow velocities covering the entire velocity range in the test section. This range was determined based on the numerical design of the compressor. The flow velocity was calculated from the measurement of the static and total pressures together with the Prandtl probe with the following equation:

$$M = \sqrt{\frac{2}{\gamma - 1} \left[\left(\frac{P_0}{P_s} \right)^{\frac{\gamma - 1}{\gamma}} - 1 \right]} \quad (4.33)$$

4.2. Inlet Stage Flow Conditions

77

As announced earlier, this calibration has to be performed at constant flow density, corresponding to the nominal rotational speed conditions. The density is given by the relation:

$$\rho = \frac{P_s}{T_s R} \quad (4.34)$$

with R the perfect gas constant and T_s the static temperature. The determination of the static temperature is based on the measurement of the total temperature and the static-to-total pressure ratio via the equation:

$$T_s = \left(\frac{P_s}{P_0} \right)^{\frac{\gamma-1}{\gamma}} T_0 \quad (4.35)$$

The first step of the calibration is the determination of the flow density under nominal conditions. Next, the compressor rotational speed is modified from 105% to 0%. At each velocity the pressure in the tank is adjusted in order to get the nominal flow density. The calibration curve is shown on Fig.4.13. This curve is then fitted, using a power law curve fit:

$$v = \left(\frac{E^2 - A}{B} \right)^{\frac{1}{n}} \quad (4.36)$$

giving the values for constants $A = 5.547993$, $B = 36.976089$ and $n = 0.370907$.

From the equation 4.30, we can see that the temperature of the flow has a direct influence on the voltage over the wire. The development presented here is valid for temperature variation lower than 5 K [34]. If the temperature variations higher, as it is the case in the R4 facility during the tests, the measured voltage E_{meas} at temperature T_{meas} has to be corrected. This correction is given by the relation:

$$E_0 = E_{meas} \sqrt{\frac{T_w - T_0}{T_w - T_{meas}}} \quad (4.37)$$

where E_0 is the corrected voltage at the reference temperature T_0 , and T_w is the temperature of the wire. The wire temperature is determined by measuring the voltage over the wire for a constant flow velocity at two temperatures 1 and 2:

$$T_w = \frac{T_2 E_1^2 - T_1 E_2^2}{E_1^2 - E_2^2} \quad (4.38)$$

4.2. Inlet Stage Flow Conditions

The importance of establishing the radial distribution of the flow characteristics is twofold. First, this distribution is used as inlet condition for the numerical simulations. This allows to certify that the simulations are performed exactly under identical flow conditions as the measurements.

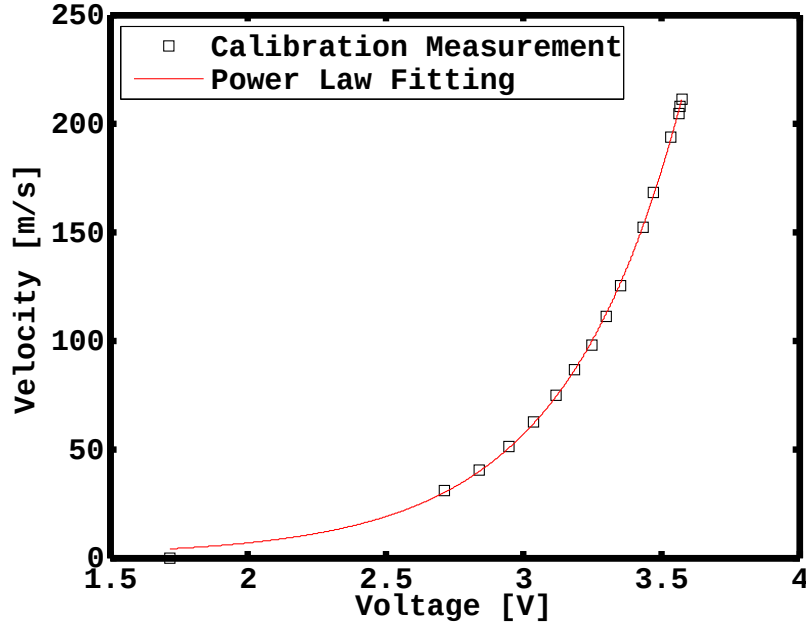


Figure 4.13.: Voltage given by a hot-wire as function of the flow velocity at constant density

Secondly, the stage inlet conditions are used in the determination of various global performance parameters of the compressor stage, like the determination of the mass flow passing through the compressor stage, or the efficiency, or else the pressure ratio of the compressor. As to the mass flow, its determination should be calculated initially from a diaphragm placed in the return loop of the R4 facility. Due to the too high pressure drop caused by the diaphragm and problems of compatibility of installation requirements with the international standards concerning the use of diaphragms, the mass flow is finally a derivative of the measurement of the radial distribution of the static- and total pressure as well as total temperature at the inlet of the compressor stage. The global pressure ratio and efficiency of the compressor stage are based on these radial distributions as well. The uncertainties on the inlet flow conditions have a direct influence on the uncertainties on the overall performance of the compressor.

The turbulence intensity is a parameter with a big influence on the numerical simulations via the turbulence model. This is used to model the unsteady motion of the flow which cannot be resolved directly because of the computer capacity limitation. The turbulence intensity Tu is given by the ratio of the root-mean-

square of the local flow velocity, v_{RMS} , to the mean value of this local velocity v_{mean} :

$$Tu = \frac{v_{RMS}}{v_{mean}}, \quad (4.39)$$

where the value of v_{RMS} over N samples is given by:

$$U_{RMS} = \sqrt{\frac{1}{n-1} \sum_1^n (v_i - v_{mean})^2}, \quad (4.40)$$

The measurement of the unsteady flow velocity is performed by means of the hot-wire probe.

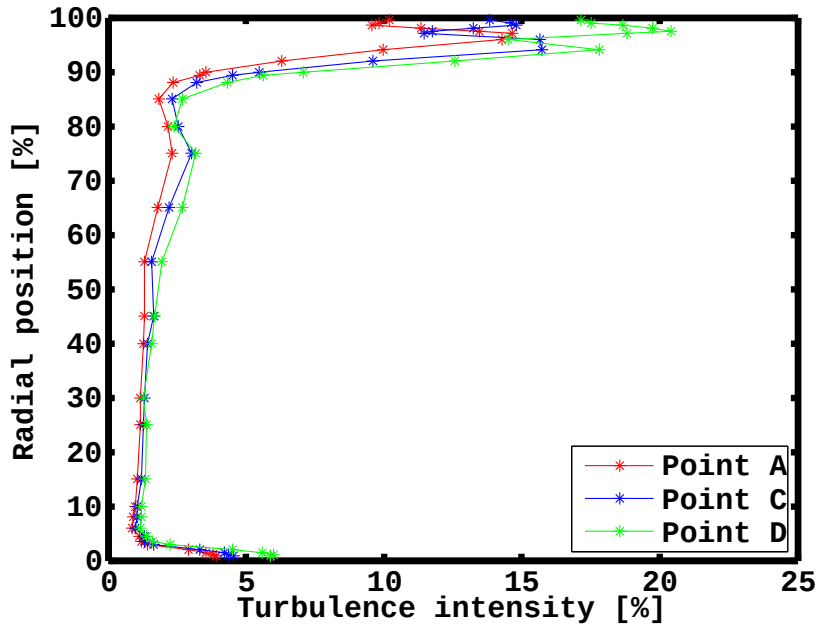


Figure 4.14.: Experimental radial distribution of the turbulence intensity at the stage inlet for the various throttling conditions under cruise pressure condition

The radial distributions of the turbulence intensity at the inlet of the compressor stage at the various throttling conditions are presented on Fig.4.14. The turbulence intensity is seen to vary between 1.25%, at the hub outside the boundary layer, and up to 2.5% at about 75 – 80% of the channel height. The turbulence level

increases from operating point A to D. This is due to the decrease of the mean value of this local velocity v_{mean} . In the tip region, very high peaks of turbulence are visible. These peaks, exceeding 10% are attributed to the artificial turbulences created by the leakage around the probe stem. This was verified numerically in the PhD research of Balazs Farkas. He has modeled the flow around the stem by a cavity in the casing and by imposing as wall condition the tip of this cavity to be an inlet. The mass flow of the leakage was adjusted by modifying the static pressure imposed at the inlet wall of the cavity. Figure 4.15 presents the radial distribution of the turbulence intensity for smooth casing wall and with the cavity for various leakage mass flows. A very high peak, detached from the wall, is visible in the case of the injection. The radial position and the magnitude of this peak is depending on the static pressure imposed at the tip wall of the cavity.

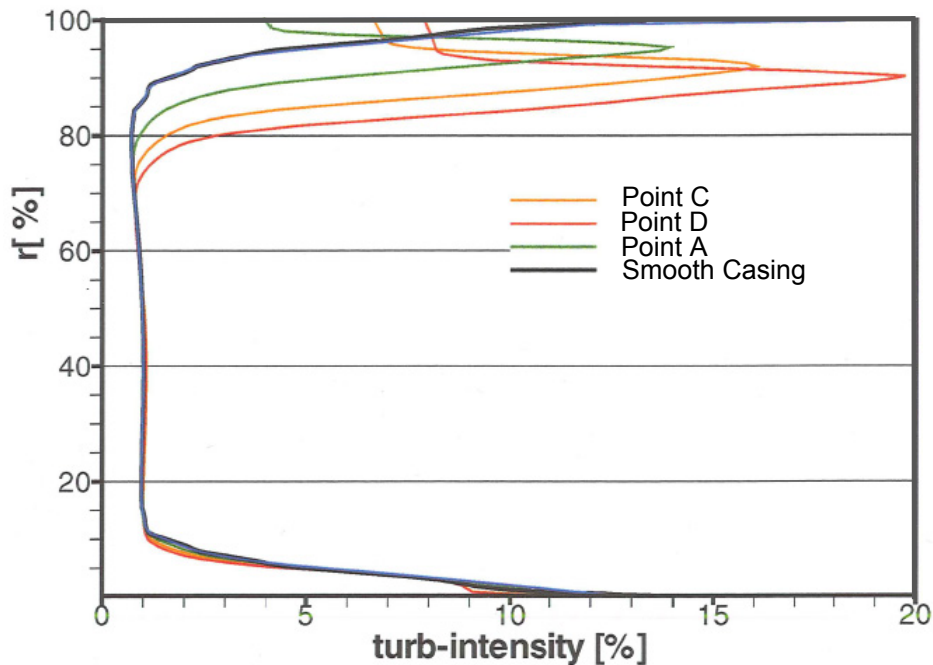


Figure 4.15.: Numerical radial distribution of the turbulence intensity at the stage inlet for the smooth casing and for the configuration with a cavity modeling the leakage around the probe stem for various leakage mass flows. Simulations with the code FINE by B. Farkas

The total pressure radial distributions are measured by means of the Kiel probes. These distributions are presented on Fig.4.16 for the various operating points under cruise conditions. On this figure, the pressure is non-dimensionalized by

the total pressure measured in the tank. Between hub and 60% of the blade height, the difference between the total pressure profiles for the various throttling conditions is lower than the uncertainty on the measurement. The measurement at the various throttling points were performed at the same circumferential position. Indeed, in 2007, Chomé [19] has shown that there exists a slight difference of about 1% between the radial distribution of the total pressure at the inlet of the stage, measured at various circumferential positions. This difference being higher than the uncertainty on the pressure measurement (about 0.1%), the difference is due to a circumferential non-uniformity in the test section. It is hence essential to perform the measurement of each flow parameter at the same circumferential position.

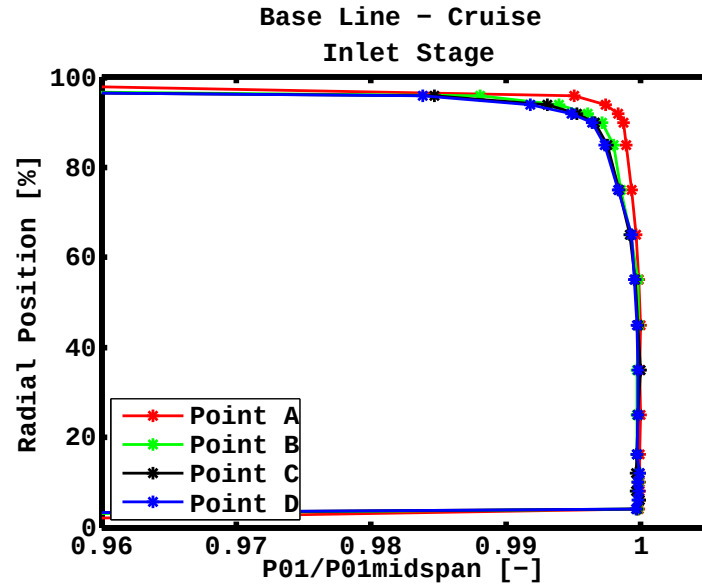


Figure 4.16.: Experimental radial distribution of the total pressure at the stage inlet non-dimensionalized by the mid-span pressure for the smooth casing configuration under cruise conditions at various operating points

Above 60%, the effect of the variation of the tip boundary layer in function of the throttling is visible. Furthermore, Fig.4.16 shows that the boundary layer at hub is smaller than at tip. The boundary layer at the hub is so thin that it is not possible to catch it due to the size of the Kiel probe. The use of the hot-wire probe is required to determine the characteristics of both boundary layers. From the measurement of the Mach number by hot-wire at the inlet of the stage, the

displacement thickness δ_d is given by the equation [52]:

$$\delta_d = \int_0^{inf} \left(1 - \frac{v_i}{v_{free}} \right) dr \quad (4.41)$$

where v_i is the flow velocity at radial position i , v_{free} is the velocity in the free stream, and r indicates the radial direction. The values of the displacement thickness for the various throttling points are given in table 4.1. The difference between

Throttling Point	Hub	Tip
A	0.1583 mm	1.0445 mm
B	0.2244 mm	1.5209 mm
C	0.3804 mm	2.3887 mm
D	0.4391 mm	3.4561 mm

Table 4.1.: Tip- and hub displacement thickness at the inlet of the compressor stage for various operating points under cruise conditions

the displacement thickness at hub and tip is obvious. This can be explained by the presence of the bulb at the inlet of the test section and by the recirculation area at the outlet of the tank. The recirculation area at the outlet of the tank is due to the enlargement of the flow channel, as presented in section 3.2, and induces an increase of the boundary layer thickness along the tip casing. In the hub region, the bulb creates a very strong acceleration of the flow, leading to a decrease of the size of the boundary layer. The enlargement of the boundary layer due to a lowering of the flow velocity explains the variation of the displacement thickness between the various throttling points as well.

The incompressible shape factor of the boundary layer SF allows determining if this boundary layer is detached or not. The factor is defined as the ratio of the displacement thickness to the incompressible momentum-loss thickness δ_m :

$$SF = \frac{\delta_d}{\delta_m} \quad (4.42)$$

and the incompressible momentum-loss thickness is given by the equation [52]:

$$\delta_m = \int_0^{inf} \frac{v_i}{v_{free}} \left(1 - \frac{v_i}{v_{free}} \right) dr \quad (4.43)$$

The values of the incompressible momentum-loss thickness and the incompressible shape factor at the inlet of the stage under cruise conditions for the various running conditions are given in the table 4.2 and 4.3 respectively.

According to Kline et al. [62], the value of the shape factor indicates the separation of the boundary layer by:

- $SF < 2.2 \Rightarrow$ Attached boundary layer,

4.2. Inlet Stage Flow Conditions

83

- $SF > 4 \Rightarrow$ Detached boundary layer,
- $SF = 2.7 \Rightarrow$ Intermittent separation,

Following these relations, the tip and hub boundary layer at the inlet of the stage under cruise conditions are completely attached for all running conditions, except the hub boundary layer at choke. At this point, the value of the shape factor indicates that the hub boundary layer is very close to be completely detached. Nevertheless, the measurements with the hot-wires did not show any back flow along the hub wall.

Throttling Point	Hub	Tip
A	0.0405 mm	0.4725 mm
B	0.1042 mm	0.7298 mm
C	0.2513 mm	1.2629 mm
D	0.3814 mm	1.9567 mm

Table 4.2.: Tip- and hub momentum-loss thickness at the inlet of the compressor stage for various operating points under cruise conditions

Throttling Point	Hub	Tip
A	3.915	2.217
B	2.153	2.084
C	1.514	1.891
D	1.151	1.766

Table 4.3.: Tip- and hub shape factor of the boundary layer at the inlet of the compressor stage for various operating points under cruise conditions

The calculation of the mass flow passing through the compressor stage is based on the radial distribution of the total pressure P_{01} from the Kiel probe, of the static pressure P_{s1} from the three-hole probe, and of the total temperature T_{01} from the shielded thermocouple probe at the inlet of the test section. Because of the geometry and size of the probes, the measurement in the core flow is limited to between 4% and 96% of the blade height. In order to obtain a complete radial distribution, the static wall pressure measured at hub and tip must be taken as total and static pressure at respectively 0 and 100%. And the total temperature at 0 and 100% of the blade height must be equal to the total temperature at respectively 4 and 96%.

Considering the total and the static pressure at radial position i , $P_{01,i}$ and $P_{s1,i}$, the Mach number at this position M_i is calculated using the equation for adiabatic

84 Chapter 4. Overall Performance of the Compressor Stage

perfect gases [101]:

$$M_i = \sqrt{\frac{2}{\gamma - 1} \cdot \left[\left(\frac{P_{01,i}}{P_{s1,i}} \right)^{\frac{\gamma-1}{\gamma}} - 1 \right]}, \quad (4.44)$$

where the adiabatic exponent γ is equal to 1.4 for dry air at 293K. Based on the gas constant of air R ($287.7 \frac{J}{kgK}$), the density ρ_i is calculated as:

$$\rho_i = \frac{P_{01,i}}{RT_{01,i}} \left[1 + \frac{\gamma - 1}{2} M_i^2 \right]^{\frac{1}{1-\gamma}} \quad (4.45)$$

Using the static temperature $T_{s1,i}$ defined by the equation:

$$T_{s1,i} = \frac{T_{01,i}}{1 + \frac{\gamma-1}{2} M_i^2} \quad (4.46)$$

The speed of sound is determined by the following equation:

$$a_i = \sqrt{\gamma R T_{s1,i}} \quad (4.47)$$

and with it the flow velocity is

$$v_i = a_i M_i \quad (4.48)$$

With R_i the radius at position i , the circular area ΔA_i is defined by

$$\Delta A_i = \pi \left(\frac{R_i - R_{i+1}}{2} \right)^2 - \pi \left(\frac{R_{i-1} - R_i}{2} \right)^2 \quad (4.49)$$

and the mass flow m_i through this area is calculated using the expression:

$$\dot{m}_i = \rho_i v_i \Delta A_i \quad (4.50)$$

The reduced mass flow at position i is calculated from the inlet conditions during the measurement at this position via the equation:

$$\dot{m}_{red,i} = \frac{\dot{m}_i \sqrt{RT_{01,i}}}{P_{01,i} D_{tip}^2} \quad (4.51)$$

This allows taking into account the variation of the inlet conditions between two measurement positions.

In order to get the global reduced mass flow at the inlet of the stage $\dot{m}_{red}$, the local reduced mass flow $\dot{m}_{red,i}$ are added up over the N radial positions:

$$\dot{m}_{red} = \sum_{i=1}^N \dot{m}_{red,i} \quad (4.52)$$

4.2. Inlet Stage Flow Conditions

85

The uncertainty on the reduced mass flow is determined using the following set of equations.

From equation 4.52 of the reduced mass flow, its uncertainty is equal to the sum of the uncertainties on the reduced mass flow at the various radial positions:

$$\delta \dot{m}_{red} = \sum_{i=1}^N \delta \dot{m}_{red,i} \quad (4.53)$$

and, from equation 4.51, the uncertainty on the reduced mass flow at point i is given by:

$$\delta \dot{m}_{red,i} = \sqrt{\left(\frac{\sqrt{RT_{01,i}}}{P_{01,i} D_{tip}^2} \cdot \delta \dot{m}_i \right)^2 + \left(\frac{\dot{m}_i \sqrt{R}}{2 \sqrt{T_{01,i}} P_{01,i}} \cdot \delta T_{01,i} \right)^2 + \left(-\frac{\dot{m}_i \sqrt{RT_{01,i}}}{P_{01,i}^2 D_{tip}^2} \cdot \delta P_{01,i} \right)^2} \quad (4.54)$$

The uncertainty on the mass flow $\delta \dot{m}_i$, based on equation 4.50, is defined by:

$$\delta \dot{m}_i = \sqrt{(v_i \Delta A_i \delta \rho_i)^2 + (\rho_i \Delta A_i \delta v_i)^2} \quad (4.55)$$

Where the uncertainty on the density $\delta \rho_i$ is given by:

$$\delta \rho_i = \sqrt{\left(\frac{\partial \rho_i}{\partial P_{01,i}} \cdot \delta P_{01,i} \right)^2 + \left(\frac{\partial \rho_i}{\partial T_{01,i}} \cdot \delta T_{01,i} \right)^2 + \left(\frac{\partial \rho_i}{\partial M_i} \cdot \delta M_i \right)^2} \quad (4.56)$$

With

$$\frac{\partial \rho_i}{\partial P_{01,i}} = \frac{1}{RT_{01,i}} \cdot \left[1 + \frac{\gamma - 1}{2} M_i^2 \right]^{\frac{1}{1-\gamma}} \quad (4.57)$$

$$\frac{\partial \rho_i}{\partial T_{01,i}} = -\frac{P_{01,i}}{RT_{01,i}^2} \cdot \left[1 + \frac{\gamma - 1}{2} M_i^2 \right]^{\frac{1}{1-\gamma}} \quad (4.58)$$

$$\frac{\partial \rho_i}{\partial M_i} = \frac{(\gamma - 1) M_i}{1 - \gamma} \cdot \frac{P_{01,i}}{RT_{01,i}} \cdot \left[1 + \frac{\gamma - 1}{2} M_i^2 \right]^{\frac{\gamma}{1-\gamma}} \quad (4.59)$$

The uncertainty on the flow velocity used in equation 4.55 is given by:

$$\delta v_i = \sqrt{(a_i \delta M_i)^2 + (M_i \delta a_i)^2} \quad (4.60)$$

where the uncertainty on the Mach number is:

$$\delta M_i = \sqrt{\left(\frac{1}{M_i \cdot \gamma \cdot P_{s1,i} \cdot \left(\frac{P_{01,i}}{P_{s1,i}} \right)^{\frac{1}{\gamma}}} \cdot \delta P_{01,i} \right)^2 + \left(-\frac{P_{01,i}}{M_i \cdot \gamma \cdot P_{s1,i}^2 \cdot \left(\frac{P_{01,i}}{P_{s1,i}} \right)^{\frac{1}{\gamma}}} \cdot \delta P_{s1,i} \right)^2} \quad (4.61)$$

and the uncertainty on the sound velocity is given by:

$$\delta a_i = \frac{\gamma R}{2\sqrt{\gamma R T_{s1,i}}} \cdot \delta T_{s1,i} \quad (4.62)$$

Finally the uncertainty on the static temperature $\delta T_{s1,i}$ is calculated from the following equation:

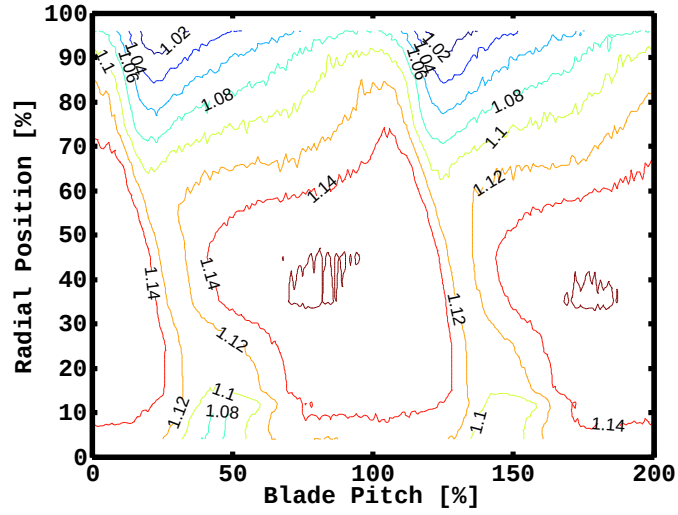
$$\delta T_{s1,i} = \sqrt{\left(\frac{1}{1 + \frac{\gamma-1}{2} M_i^2} \delta T_{01,i} \right)^2 + \left(-\frac{T_{01,i} (\gamma-1) M_i}{\left(1 + \frac{\gamma-1}{2} M_i^2\right)^2} \delta M_i \right)^2} \quad (4.63)$$

The calculation of the uncertainty of the total and static pressure, and of the temperature is given in the previous section. This procedure and the values of the uncertainties presented above give an uncertainty on the mass flow under cruise conditions from 0.8% at operating point A to 1.5% at operating point D.

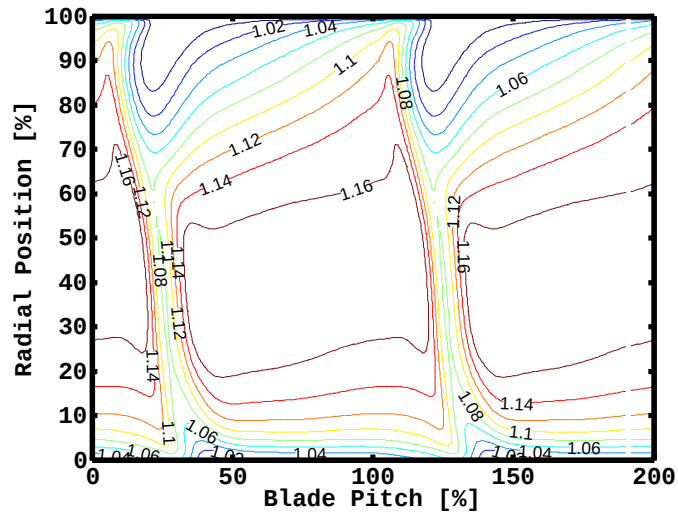
4.3. Global Stage Performance for the Smooth Casing Configuration

The global performance of the compressor stage is presented as the total-to-total pressure ratio, the static-to-total pressure ratio and the efficiency in function of the reduced mass flow. The values of these parameters were established at the four operating points presented in section 3.5, corresponding to an operating range from choke (point A) to near stall (point D). The limitation of the determination of the parameters to four points comes from the very long time required for the measurements necessary for this determination. The global performance of the compressor stage was investigated at two Reynolds numbers, representing the cruise and the take-off conditions of the real engine. The Reynolds number at cruise calculated from the measurement in the test section is 220000, which is similar to the value given by our partner Techspace Aero for the design of the compressor. Under take-off conditions, the Reynolds number from the measurement is 560000. This is lower than the Reynolds number from the design of the compressor, which is 700000. The difference comes from the limitation of the electric motor driving the compressor stage. As explained in section 3.1, the maximum power of the motor is not sufficient to drive the rotor under a pressure of 3.5 *bars* absolute, corresponding to the take-off conditions. An intermediate pressure of 2.5 *bars* absolute, sustained by the motor, was chosen in order to study the effect of the Reynolds number on the global performance of the compressor stage.

The pressure ratios presented in this report are the total-to-total pressure ratio $\frac{P_{04}}{P_{01}}$ and the static-to-total pressure ratio $\frac{P_{s4}}{P_{01}}$. Both are based on the mass flow average of the pressure radial distribution at the inlet of the stage P_{01} , and of pressure map over two pitches at the outlet of the stage P_{s4} and P_{04} . The calculation



(a) Experimental pressure ratio



(b) Numerical pressure ratio - FINE

Figure 4.17.: Experimental and numerical (FINE) total pressure map at the outlet of the stage at design operating point under cruise conditions.

of the uncertainties on these pressure ratios are based on the following equation:

$$\delta \left(\frac{P_a}{P_b} \right) = \sqrt{\left(\frac{1}{P_b} \cdot \delta P_a \right)^2 + \left(-\frac{P_a}{P_b^2} \cdot \delta P_b \right)^2} \quad (4.64)$$

This equation gives values of uncertainty on the pressure ratios of about 0.6% under cruise conditions.

The radial distribution of the flow characteristics at the inlet of the stage is set out in the previous section. The total and static pressures at the outlet of the stage are measured by means of a Kiel probe and a three-hole probe respectively. The measurements of these pressures are performed over two stator pitches, via the rotation of the stators as presented in section 3.2, and over nearly the entire blade height by means of carriage systems. In the radial direction, the measurements were performed at 17 positions from 4% to 96% of the blade height. At every radial position, the flow characteristics were acquired continuously during the rotation of the stators. The experimental and numerical total pressure maps at the outlet of the stage at design operating point under cruise conditions are presented on Fig.4.17. The experimental map is obtained by the interpolation of the data measured along the radial and circumferential axes. These plots show that the low pressure region extending from hub to tip, corresponding to the wake of the stator blades, is very well predicted by the CFD. The numerical pressure distribution from the tip region to 50% of the blade height presents a good agreement with the experimental data. In the hub region, the experimental pressure map shows that the stator suffers some hub corner stall which is not well predicted numerically. This is due to the influence of the leakage flow entering the blade passage between rotor and stator, the seal leakage flow. Indeed, this leakage flow is not taken into account in the numerical simulations and might induce some changes in stator inlet conditions favoring hub corner stall.

The similitude between the results of the numerical simulations and the experimental data in terms of total pressure at the outlet of the stage is highlighted on Fig.4.18. On this figure, the circumferential distribution over two stator pitches of the total pressure at 90%, 50% and 10% of the blade height is presented. Nevertheless, the experimental curves always show lower maxima and higher minima due to the spatial resolution of the probe. It must be recalled that the stator pitch is 17.3mm at the tip and although the Kiel probe head is only 1.5mm in diameter, this correspond to 8.5% of the pitch, which again underlines the challenge of measurements in machines of small dimensions. The non-negligible size of the instrumentation leads to a spatial averaging of the measured value. This induces a reduction of the maxima and of the gradients. Whereas the wake position is well predicted at the tip, a gradual shift of the wake position takes place towards the hub. This is again due to the under-estimation of the stator hub corner stall due to disregarding the seal leakage flow. This is confirmed by the absolute exit angle α_4 showing more underturning in the experiments. The superposition of the circumferential distribution at the three radial positions on the same figure allows

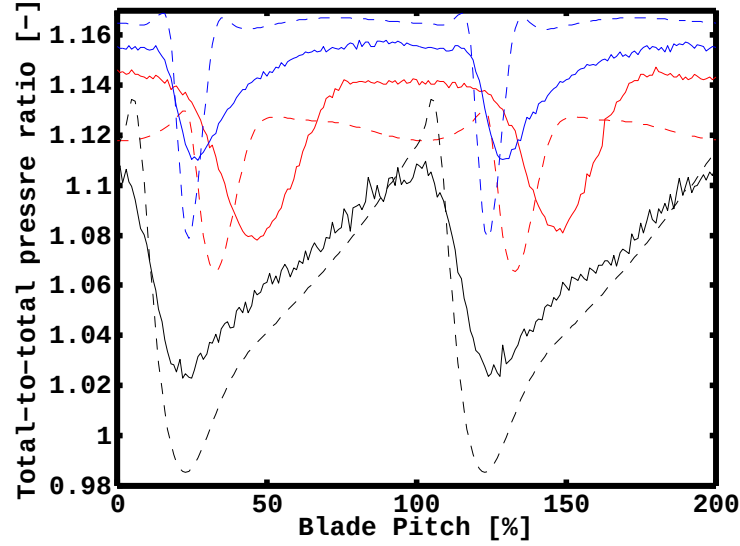


Figure 4.18.: Experimental (full line) and numerical (FINE - dashed line) circumferential distribution of the total-to-total pressure ratio P_{04}/P_{01} under cruise conditions at the design operating point at 10% (in red), 50% (in blue) and 90% (in black) of the blade height

to compare the total pressure deficit in the wake at midspan versus the wake at hub and the tip leakage vortex loss core at the tip.

The experimental total-to-total pressure ratios at nominal speed under cruise and take-off conditions are presented on Fig.4.19 in function of the reduced mass flow, with the error bars from the uncertainty calculations, and compared to the results of the numerical simulations under cruise conditions. The numerical simulations are performed with two codes and various turbulence models, as presented in chapter 3.4. On the one hand, our partner CENAERO used the elsA code with three turbulence models, $k - \epsilon$, $k - \omega$ and $k - l$. On the other hand, B. Farkas simulated the compressor with FINE and with the $k - \epsilon$ turbulence model at VKI. The mesh considered in the simulations with elsA for the various turbulence models is the same, but this mesh is different in the simulations with FINE. The speed-lines presented on the figure show a peak corresponding to the design operating point. This indicates the development of regions with a high velocity defect [61] before the compressor reaches the stall point. These large regions of blockage induce a reduction of the effective flow passage area, leading to a reduction of the total-to-total pressure ratio. This blockage of the flow is due to the development of the secondary flows in the compressor stage. These secondary

flows are the tip leakage vortex in the tip region and the hub corner separation at the hub, as presented in the literature review of this thesis. The development of the large blockage region before the stability limit is characteristic of highly hub loaded compressors. Indeed, in these compressors, the high load in the hub region enhances the development of the hub corner separation in hub corner stall, leading to a large blockage. The pressure deficient area caused by the blockage can be observed by the measurement of the unsteady pressure downstream the rotor. The results of these measurements are presented in the next chapter.

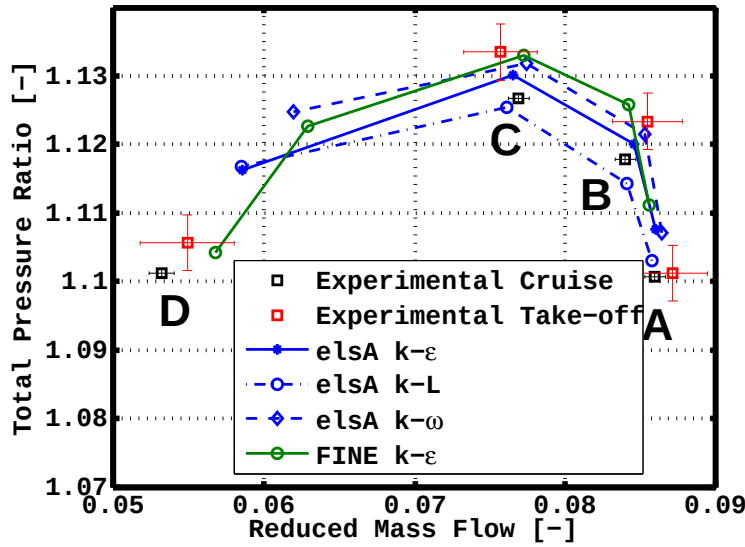


Figure 4.19.: Experimental total-to-total pressure ratio under cruise and take-off conditions at nominal speed, and numerical total-to-total pressure ratio at nominal speed under cruise conditions in function of the reduced mass flow

The comparison between the numerical results and the experimental measurements shows the inability of the CFD to simulate the compressor stage near the stability limit. Indeed, near stall the computation starts to show oscillations in the convergence history or does not converge at all for higher throttling. This is caused by the fact that under this condition the flow is mainly driven by unsteady effects, which are hard to capture by steady RANS computations. Under all operating conditions, except near stall, the reduced mass flow is very well predicted by the CFD. In terms of pressure ratio, it is over-predicted in the numerical simulations. As shown previously, the numerical simulations do not take into account the seal

4.3. Global Stage Performance for the Smooth Casing Configuration

91

leakage flow. This leads to an over-prediction of the total pressure at the outlet of the stage, meaning an over-estimation of the total-to-total pressure ratio of the compressor stage.

Secondly, the comparison of the results of the simulation performed with the elsA code, with various turbulence models, presents high discrepancies. Regarding the pressure ratio, variations between the results of about 0.08 are observed at design operating point. Compared to the experimental results, the smallest difference with respect to the measured pressure ratio is obtained by using elsA with the $k-l$ model, and the highest with the $k-\omega$ model. A comparison of the efficiency given by the various models is necessary to choose the most appropriate model.

The stall margin SM of the compressor stage can be determined from the total-to-total pressure ratio and the reduced mass flow at the design point (Point C) and at the last stable point before stall (Point D). The equation of the stall margin is given by Cumpsty [22]:

$$SM = 1 - \left(\frac{\Pi_{Design} \dot{m}_{red,Stall}}{\Pi_{Stall} \dot{m}_{red,Design}} \right) \quad (4.65)$$

where Π is the total-to-total pressure ratio and $\dot{m}_{red}$ the reduced mass flow as defined above. The experimental measurements give a stall margin of $SM = 29.2\%$.

The effect of the Reynolds number on the total-to-total pressure ratio is shown on Fig.4.19 as well. Firstly, we can observe that the uncertainty on the performance of the compressor stage at high Reynolds number are much higher than under cruise conditions. This comes from the increase of the mean pressure in the loop and the pressure difference between upstream and downstream the stage. This leads to the use of pressure transducers with higher pressure range, and thus higher measurement uncertainty, especially for the measurement of the settling chamber pressure. Experimentally, a higher Reynolds number leads to a higher pressure ratio. This is explained by the fact that transition of the boundary layer occurs earlier at high Reynolds. This leads to decreasing the detachment of the flow from the blades [91], inducing a decreasing of the blockage in the flow passage. This allows an increase of the pressure ratio performed by the compressor stage. Concerning the effect of the Reynolds number on the reduced mass flow and on the stability range of the compressor, it is very difficult to conclude from the observation of the results because the near stall point at low Reynolds number stays within the uncertainty margin of the high Reynolds number point.

Figure 4.20 shows the static-to-total pressure ratio under cruise conditions. These results show that the stability limit is reached at or very close to the zero-slope point of the curve. This behavior is typical of a compressor presenting a modal stall inception process [106]. Camp and Day [15] found that this inception mode is associated with a flow separation near the hub. This can be related to the observation of the presence of a maximum in the total-to-total pressure ratio in function of the reduced mass flow due to a large blockage at the hub. A detailed study of the compressor stall inception mode is given in chapter 6.

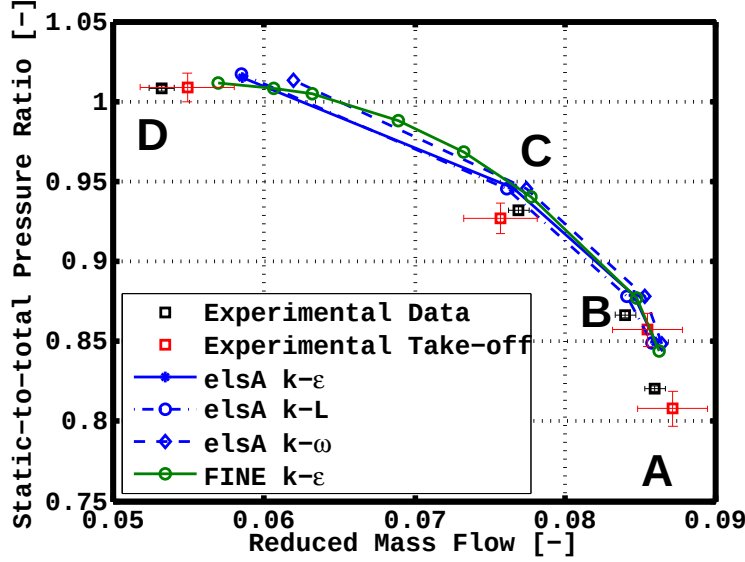


Figure 4.20.: Experimental static-to-total pressure ratio at nominal speed under cruise and take-off conditions, and numerical static-to-total pressure ratio at nominal speed under cruise conditions

The calculation of the aerodynamic efficiency is based on the mass flow average of the total pressure and temperature radial distributions at the inlet of the stage, P_{01} and T_{01} , and on the mass flow average of the total pressure and temperature at the outlet of the stage, based on the measurement map over two pitches, P_{04} and T_{04} . And in order to minimize the influence on the efficiency of the inlet conditions between the various tests, the temperature at the inlet and at the outlet of the stage are acquired simultaneously. The efficiency is given by the equation:

$$\eta = \frac{\left(\frac{P_{04}}{P_{01}}\right)^{\frac{\gamma-1}{\gamma}} - 1}{\frac{T_{04}-T_{01}}{T_{01}}} \quad (4.66)$$

The uncertainty on the efficiency calculation is given by the equation:

$$\delta\eta = \sqrt{\left(\frac{\partial\eta}{\partial P_{01}} \cdot \delta P_{01}\right)^2 + \left(\frac{\partial\eta}{\partial P_{04}} \cdot \delta P_{04}\right)^2 + \left(\frac{\partial\eta}{\partial T_{01}} \cdot \delta T_{01}\right)^2 + \left(\frac{\partial\eta}{\partial T_{04}} \cdot \delta T_{04}\right)^2} \quad (4.67)$$

4.3. Global Stage Performance for the Smooth Casing Configuration

93

Where the partial derivatives are given by:

$$\frac{\partial \eta}{\partial P_{01}} = \frac{-\frac{P_{04}}{P_{01}^2} \cdot \frac{\gamma-1}{\gamma} \cdot \left(\frac{P_{04}}{P_{01}}\right)^{-\frac{1}{\gamma}}}{\frac{T_{04}-T_{01}}{T_{01}}}, \quad (4.68)$$

$$\frac{\partial \eta}{\partial P_{04}} = \frac{\frac{\gamma-1}{\gamma P_{01}} \cdot \left(\frac{P_{04}}{P_{01}}\right)^{-\frac{1}{\gamma}}}{\frac{T_{04}-T_{01}}{T_{01}}}, \quad (4.69)$$

$$\frac{\partial \eta}{\partial T_{01}} = \frac{\frac{T_{04}}{T_{01}^2} \cdot \left[\left(\frac{P_{04}}{P_{01}}\right)^{\frac{\gamma-1}{\gamma}} - 1\right]}{\left(\frac{T_{04}-T_{01}}{T_{01}}\right)^2}, \quad (4.70)$$

$$\frac{\partial \eta}{\partial T_{04}} = \frac{-\frac{1}{T_{01}} \cdot \left[\left(\frac{P_{04}}{P_{01}}\right)^{\frac{\gamma-1}{\gamma}} - 1\right]}{\left(\frac{T_{04}-T_{01}}{T_{01}}\right)^2}. \quad (4.71)$$

The uncertainty on the efficiency determined following this procedure is from 2.8% at choke to 3.4% near stall.

The experimental non-dimensionalized total-to-total efficiency of the compressor stage in function of the reduced mass flow is given on Fig.4.21 under cruise and take-off conditions. The numerical efficiencies under cruise conditions are added on this figure for the comparison. The efficiency at the various operating points is non-dimensionalized by the experimental efficiency at design point under cruise conditions. The total-to-total efficiency is well predicted up to design point, with a slight under-estimation by the simulations considering the error bars. Growing discrepancies appear as throttling is further increased. This is again due to the relative inability of the CFD code to predict unsteady phenomena.

The under-estimation is explained by the difference between the geometry considered for the mesh of the numerical simulations and the real geometry of the test section. The geometry used for the simulations is the ideal geometry from the design of the compressor. The real geometry present in the test section has been altered by the manufacturing and by the assembly.

An example of the geometry difference is the hub blade fillets or grooves. These come from the manufacturing procedure and is not simulated. In the case of the rotor blades, they were manufactured by machining. The process leads to the presence of a fillet at the hub of the blades. The numerical simulations do not take this into account. In 2010, Goodhand and Miller [41] found that removing the fillet of the stator blades of a low-speed single stage compressor leads to an increase of the hub loss of 18%. This detrimental effect may be more important in case of a reversal flow within the separation. Furthermore, they show a non-negligible impact of the radius of the blade fillet on the hub losses. For the stator

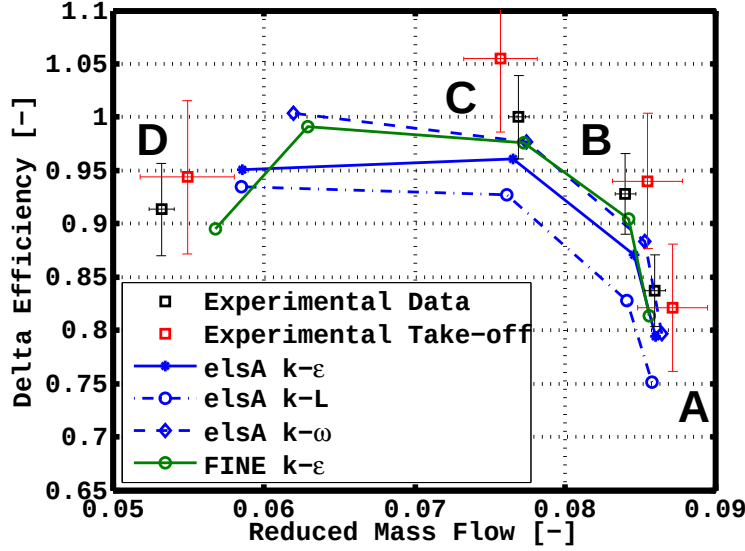


Figure 4.21.: Experimental total-to-total efficiency at nominal speed under cruise and take-off conditions, and numerical total-to-total efficiency at nominal speed under cruise conditions

blades, the airtightness was provided by a joint of silicone. A bad injection of this silicone has led to the presence of a groove around the blades at the hub, as shown on Fig. 4.22. The exact effect of this groove on the losses has not been studied.

A second geometrical parameter that can explain the under-estimation of the efficiency, is the tip gap over the rotor blades. In the CFD, the nominal tip gap of 0.3mm is considered. In section 3.2, it was observed that the experimental tip gap was lower than this value, even for one blade. According to Tan [106], up to a critical point, the reduction of the tip gap leads to an increase of the efficiency.

The comparison of the results from the three turbulence models used for the simulations with the elsA code shows a difference between the turbulence models in terms of efficiency of about 5% at design point, which is considerable. The $k-l$ model, which gave the results the closest to the experimental data in terms of pressure ratio, gives the highest difference between the measured and predicted efficiency. The $k-\epsilon$ model actually yields good results both for efficiency and pressure ratio prediction. This model was chosen for the subsequent simulations, performed with the elsA code.

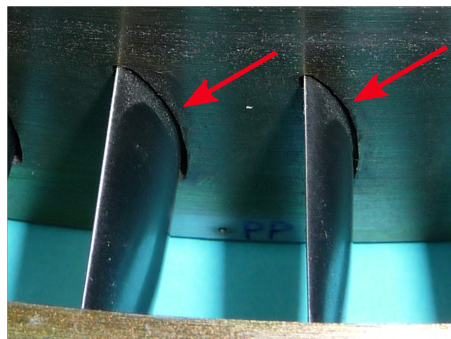
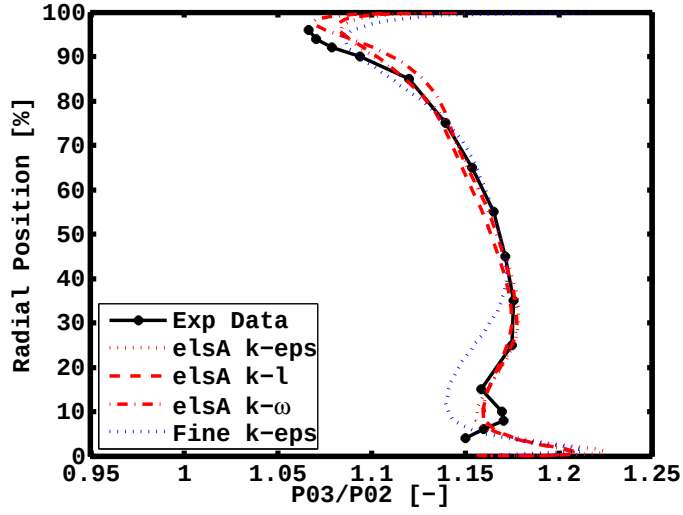


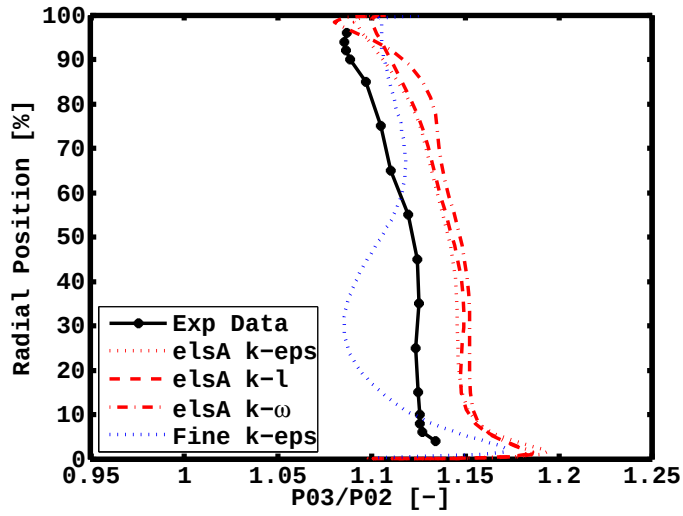
Figure 4.22.: Groove around the hub of the stator blades

4.4. Rotor Performance

The rotor performance of the compressor stage is presented on Fig.4.23 and on Fig.4.24 in terms of total-to-total pressure ratio (P_{03}/P_{02}) and temperature ratio (T_{03}/T_{02}) for the design operating point (Point C) and the near stall operating point (Point D) under cruise conditions. The other operating points are presented in Brouckaert et al. [13]. For both ratios, the measurements upstream and downstream the rotor were performed during two different tests. The non-dimensionalization together with the simultaneous inlet stage conditions allows taking into account variations between the conditions of the various tests. All the probes used for these measurements were aligned with the leading edge of the IGV. This ensures that the measurements are not performed in the wake of the IGV. At design point, the highly hub loaded distribution characteristics of the booster stage is evident. The pressure and temperature ratios are very well simulated by all the CFD codes and the turbulence models in terms of mean level of the amplitude as well as in terms of gradient. In the hub region, some discrepancies can be found between experimental and CFD results. This is due to the fact that the seal leakage flow, resulting from the gap between the fixed and mobile part of the compressor, is not considered in the numerical simulations. The losses at the rotor hub are over-predicted by the CFD FINE code. The very good agreement between the experimental and numerical rotor performance has to be compared with the discrepancies existing in the performance of the entire stage between the CFD and the experimental results. This shows that these discrepancies may be explained by the presence of mixing plane existing between the fixed and rotating parts in the simulations. These mixing planes lead to the non-consideration of the rotor-stator interaction. A comparison with unsteady simulations would be required to evaluate this effect. A second explanation is the inability of the CFD codes to predict the performance of the stators. The pressure ratio between upstream and

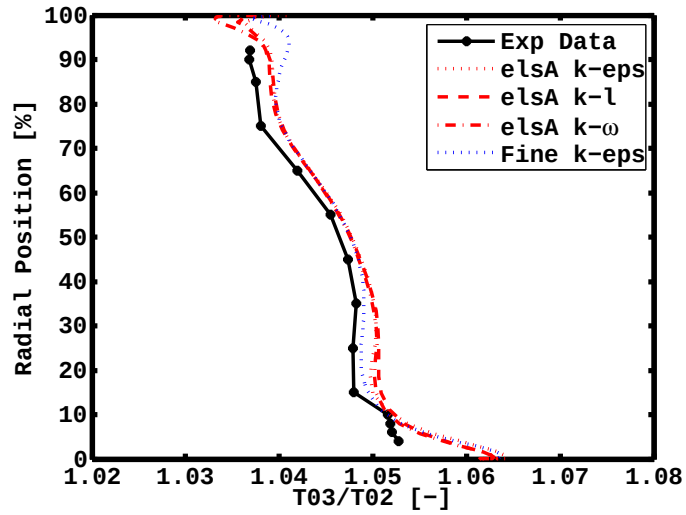


(a) Design operating point (Point C)

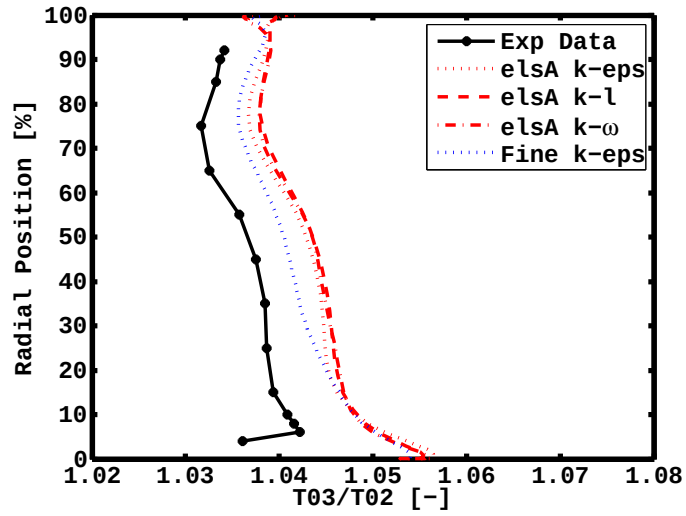


(b) Near stall operating point (Point D)

Figure 4.23.: Radial distributions of the experimental and numerical total-to-total pressure ratios of the rotor under cruise conditions at design- (a) and near stall (b) operating points.



(a) Design operating point (Point C)



(b) Near stall operating point (Point D)

Figure 4.24.: Radial distributions of the experimental and numerical total-to-total temperature ratios of the rotor under cruise conditions at design- (a) and near stall (b) operating points.

downstream the IGV is presented on Fig.4.25. On this figure, we can observe differences between the experimental and the numerical results. First of all, the mean level of the pressure ratio is lower in the CFD. This can be explained by the fact that the experimental distribution corresponds to a single circumferential position out of the wake of the IGV. The numerical results are the mass flow average of the distribution over the entire pitch, including the wake. The second point is the higher thickness of the boundary layer both at hub and at tip in the CFD. Because the inlet conditions of the numerical simulations are the measured distributions, it means that the CFD predicts an enlargement of the boundary layer through the IGV. This enlargement is not observed in the measurements. This can explain the variation between the numerical and experimental global performance of the stage.

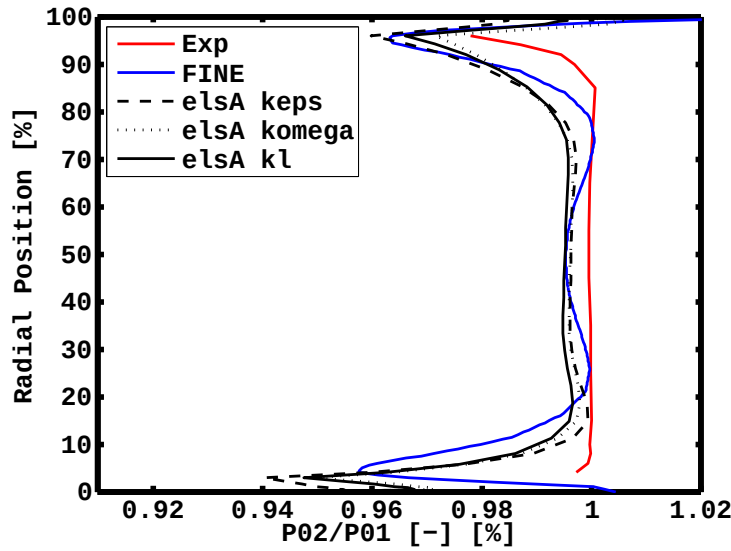


Figure 4.25.: Radial distributions of the experimental and numerical total-to-total pressure ratios of the IGV under cruise conditions at near stall operating points.

At near stall operating point, an important shift of the overall level exists between the experimental results and the numerical simulations, both in the total pressure- and temperature distributions. This is caused by the inability of the CFD codes to simulate the flow at a mass flow corresponding to the experimental near stall conditions. In the overall stage performance presented in the previous section, the last stable points in the numerical results correspond to a higher mass flow than for the experimental results, and this for all the codes and turbulence

models. However, the gradient of the pressure- and temperature ratios are well predicted. The over-prediction of the losses in the hub region by the FINE software is visible as well.

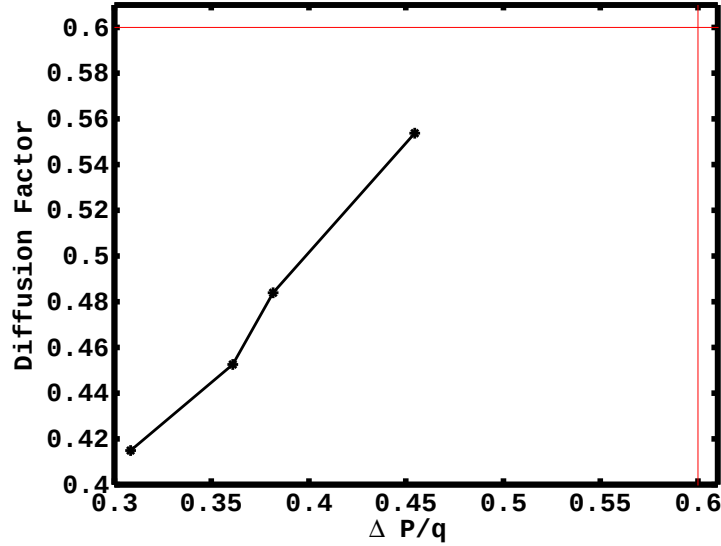


Figure 4.26.: Experimental diffusion factor of the rotor alone versus the non-dimensionalized pressure rise under cruise conditions over the entire operating range

In 1979, Greitzer et al. [43] defined a criterion for the effectiveness of a casing treatment on the stability limit of a compressor stage. This criterion is based on the diffusion factor, as defined by Lieblein [73], and on the non-dimensionalized pressure rise $\Delta P/q$. The diffusion factor is a parameter giving a theoretical limit of the deceleration before a separation of the boundary layer at the suction surface. It is given by the relation:

$$DiffusionFactor = 1 - \frac{w_3}{w_2} + \frac{\Delta v_u}{2\sigma w_2} \quad (4.72)$$

In this equation, w_2 and w_3 are the relative velocity, respectively upstream and downstream the rotor, Δv_u is the difference of the circumferential component of the absolute velocity, and σ is the rotor solidity. The relative velocity is determined from the absolute velocity and the flow angle, given by the three-holes probe, and the rotor rotational speed from the torque meter. In the Greitzer approach, the diffusion factor is related to the pressure rise, non-dimensionalized by the relative

dynamic pressure at the outlet of the rotor given by :

$$q = \frac{1}{2} \rho w_2^2 \quad (4.73)$$

where ρ is the density of the flow. The experimental diffusion factor versus the non-dimensionalized pressure rise of the compressor stage under cruise conditions is presented on Fig.4.26. Greitzer defined the stability limit for the two parameters at 0.6 and showed that the type of stall depends of the parameter reaching this value. If a compressor reaches the stability limit of 0.6 with the diffusion factor, this compressor will present a blade stall. On the other hand, if the non-dimensionalized pressure rise reaches 0.6 before the diffusion factor, the compressor will present a wall stall. In our case, the rotor row seems to reach the limit for the diffusion factor first, meaning a blade stall. From their work on this criterion, related to the effectiveness of a casing treatment on the stability limit, Greitzer et al. conclude that the casing treatment has a very low impact on the global performance of a compressor presenting a blade stall. This was confirmed more recently by the work of Houghton [54].

4.5. Effects Of The Casing Treatment On The Overall Performance

The effect of a circumferential groove casing treatment on the global performance of the compressor stage has been investigated. The circumferential groove has a depth of 1 mm with an aspect ratio of 1/3 and it is located at 15% of the axial chord, downstream the leading edge of the rotor blades. A complete description of the casing treatment is given in section 3.6.

The overall stage performance of the compressor stage in terms of total pressure ratio and relative efficiency change versus reduced mass flow under cruise conditions for the smooth casing and casing treatment configurations is presented on Fig.4.27. The experimental data for the configuration with smooth casing and with casing treatment are presented with the corresponding error bars and are compared to the results from the numerical simulations performed with the FINE and elsA codes.

Experimentally, except under choke conditions (point A), the groove slightly improves the total-to-total pressure ratio. But this improvement stays close to the uncertainty range [108]. Slight changes in efficiency are noticeable but remain within the uncertainty range of the measurements. Near stall (at point D), the configuration with casing treatment shows an increase of 3% of the mass flow, 1% of total-to-total pressure ratio and 2% of efficiency. This increase of mass flow could be interpreted as a reduction of the stability margin of the compressor. Equation 4.65 of the stall margin gives for the casing treatment configuration a margin of $SM = 28\%$. But in this case, as it will be shown later, the increase of

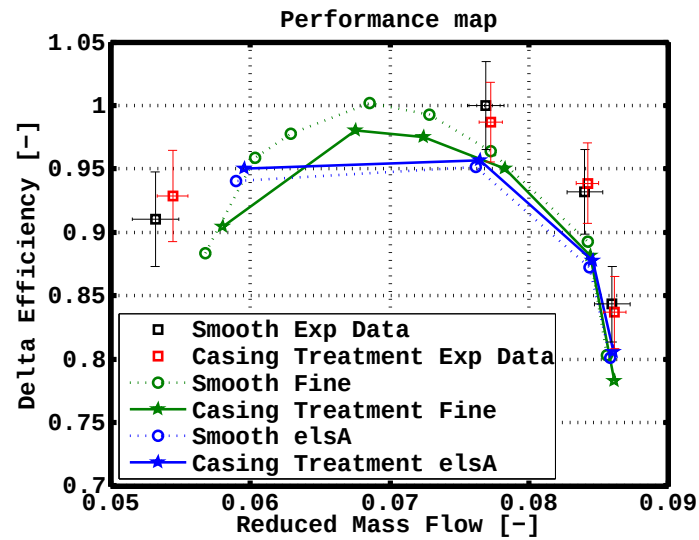
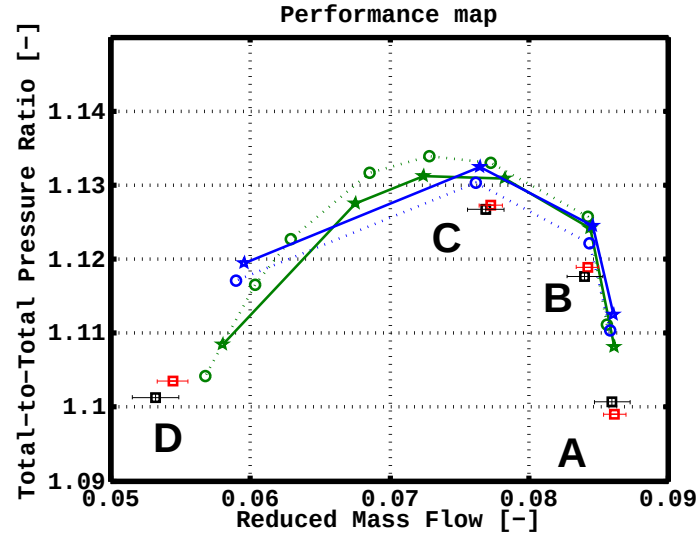


Figure 4.27.: Experimental and numerical total pressure ratio and relative efficiency versus the reduced mass flow under cruise conditions for the smooth casing and casing treatment configurations.

the performance at near stall operating point indicates a reduction of the blockage caused by the circumferential groove.

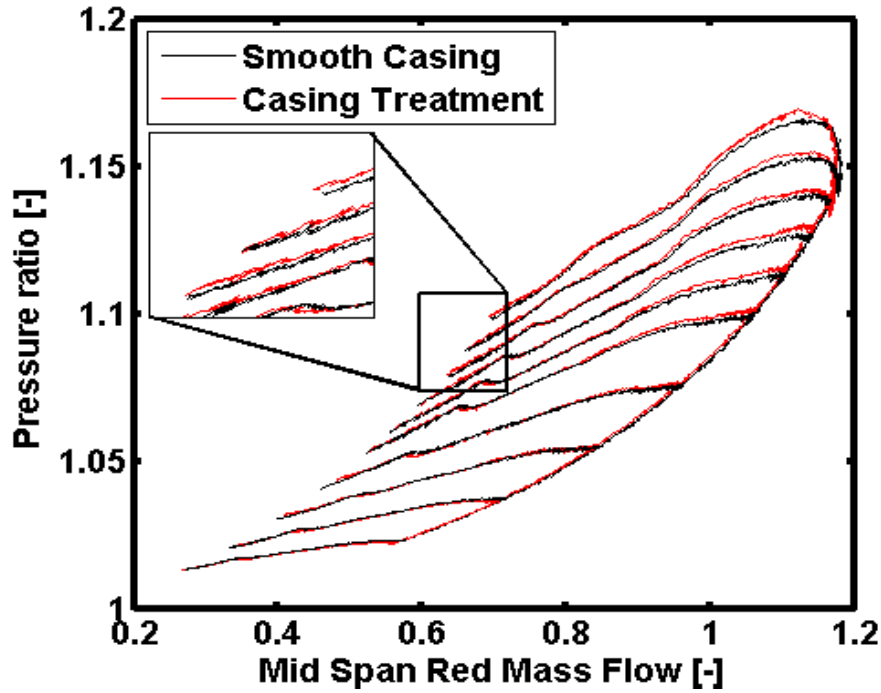


Figure 4.28.: Experimental total-to-total pressure ratio at mid-span iso-speed versus the reduced mass flow for the smooth casing and the casing treatment configurations

The numerical simulations of the compressor with and without casing treatment present a contradictory conclusion about the effect of the groove on the performance. Indeed, in the simulations with FINE, the presence of the casing groove induces a decrease of both pressure ratio and efficiency. On the contrary, elsA predicts an increase of these parameters with the casing treatment. This difference of behavior evidences the influence of the CFD code (mainly through the turbulence model, but also the mesh density or topology) on the numerical results. It underlines once more the fact that great care is required when making engineering decisions based on CFD. The inability of the steady RANS-based CFD models to predict the stall margin improvement generated by casing grooves was already pointed out by Houghton [55].

In order to investigate the influence of the treatment on the stability limit at

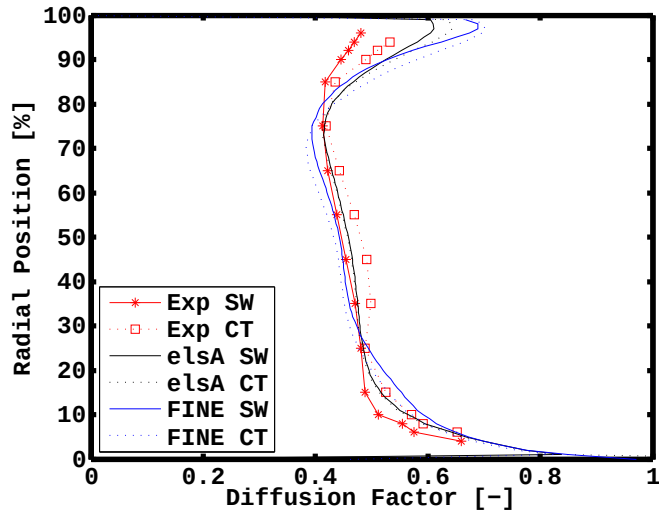
off-design, the stage inlet and outlet midspan total pressure were acquired during throttle valve closing until stall at various rotational speeds. This experiment was performed for the two casing configurations. The mid-span total-to-total pressure ratio iso-speed lines are presented on Fig.4.28. The iso-speed lines correspond to a rotational speed of 105%, 100%, 95%, 90%, 85%, 80%, 70%, 60%, 50% and 40% of the nominal rotational speed. At design and off-design, a slight improvement in pressure ratio is observed, but the stall margin of the compressor remains unchanged.

The effect of the circumferential groove casing treatment on the rotor performance was investigated by the measurement of the flow characteristics up- and downstream the blade row [108]. The influence of the groove on the rotor flow turning in the tip region was found negligible. On the other hand, the presence of the circumferential groove casing treatment leads to an increase of the diffusion factor at design and near stall operating points. This observation is confirmed by the numerical calculations performed by elsA. The radial distribution of the experimental and numerical diffusion factors at these two operating points are presented on Fig.4.29. Furthermore, the casing treatment induces an important redistribution of the loading with an increase of the diffusion factor between 80 and 100% of the blade height. The increase of the diffusion factor shows a decrease of the stall margin in the tip region. This effect is not visible on the global stall limit due to the hub critical character of this compressor.

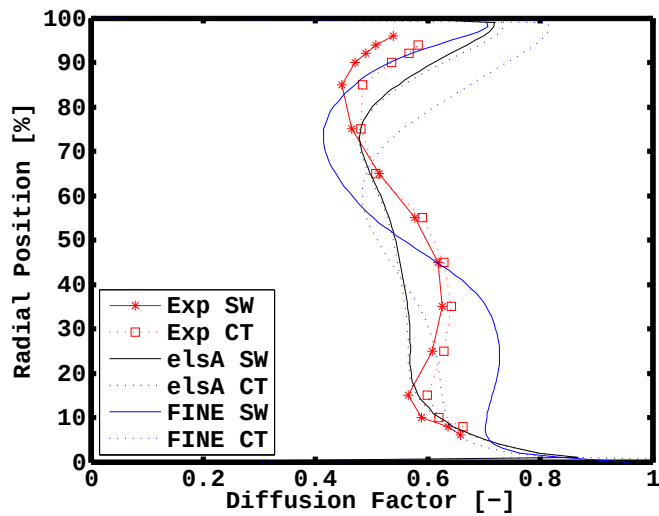
4.6. Conclusion

This chapter presented the overall performance of a single stage low pressure axial compressor. The design of this compressor stage is representative of the first stage of the low pressure compressor of the new contra-rotating turbofan engine architecture. This design is characterized by a low aspect ratio and unusually high loading- and flow coefficients. The experimental performance is compared to numerical simulations performed with various CFD codes and turbulence models.

The determination of the overall performance of the compressor stage is based on the steady characteristics of the flow, as the total and static pressure, the total temperature, the flow angle and the velocity. The steady instrumentation used for the measurement of these characteristics is detailed in this chapter as well as the uncertainties associated. The uniformity of the test section has been verified through the measurement of the wall static pressure at various axial and circumferential positions. The radial distribution of the static and total pressure has been determined at various axial positions by means of, respectively, a Kiel probe and a three-hole probe. This radial distribution does not cover the entire blade height because of the size of the probes. The distributions are completed by the values of both wall static pressures measured by pressure taps. The uncertainty calculation gives a value on the total pressure of about 0.04%, corresponding to 0.4 *mbar* under cruise conditions. The calibration of all the pressure sensors has



(a) Design operating point



(b) Near stall operating point

Figure 4.29.: Radial distribution of the experimental and numerical diffusion factor under cruise conditions for the smooth casing and casing treatment configurations at design (a) and near stall (b) operating points.

been performed daily in-situ at two pressure levels. A three-hole probe was used for the determination of the flow velocity and angle via a preliminary calibration in terms of angle and Mach number, which has been performed in the VKI C4 facility. The uncertainty on the static pressure, the flow angle and the Mach number determined by the three-hole probe is respectively about 0.4 *mbar*, 0.2° and 1.5%. The total temperature was measured by means of a thermocouple probe. The values from this probe have been corrected in function of the flow velocity via the recovery factor in order to take into account the velocity error. This correction is an important source of error, leading to the a high uncertainty of 0.3 *K* on the temperature measurement. These values lead to an unceratinty between 0.8% and 1.5% on the mass flow, between 2.8% and 3.4% on the efficiency and about 0.6% on the total-to-total pressure ratio.

The flow conditions at the inlet of the compressor stage were used for the determination of the overall performance of the compressor as well as inlet conditions for the numerical simulations. They had thus to be carefully determined. The radial distribution of the turbulence at the inlet of the compressor is between 1.25% at the hub up to 2.5% at 80% of the blade height. In the tip region, the turbulence presents a very high peak caused by the leak around the probe stem. This was confirmed by numerical simulations considering the leak via a cavity in the casing wall. The radial distribution of the total pressure at the inlet of the compressor stage shows a displacement thickness at tip seven times higher than at hub. This large difference in the size of the boundary layer is caused by the geometry of the inlet part of the test section.

The radial and circumferential distributions of the total pressure at the outlet of the compressor stage show a large low pressure regions along the suction side of the stator blades, corresponding to the corner vortex secondary flows. These distributions are well simulated by the software FINE, particularly in the tip region. The comparison of the experimental and numerical circumferential distributions of the total pressure shows a gradual shift of the position of the wake of the blade going towards the hub. This is due to the under-estimation of the hub corner stall in the CFD results by neglecting the seal leakage flow occurring in this region. Furthermore all the pressure extrema are attenuated in the experimental results due to the area averaging introduced by the geometrical extension of the probe.

The total-to-total pressure ratio in function of the reduced mass flow presents a peak characteristics of a hub loaded compressor. The decrease of the pressure ratio with the mass flow is caused by a pressure defect in the hub region. This is due to the development of the hub corner stall before the stability limit, creating high blockage in the flow passage. The experimental stall margin determined from this characteristic is 29.2%. The stability limit is reached for a nearly zero slope of the static-to-total characteristic. According to Tan [106], this indicates a compressor presenting a modal stall inception process, which can be related to large flow separation at the hub [15]. This confirms the presence of a hub corner stall near the stability limit of the compressor. The investigation of the effect of the Reynolds number shows that a higher Reynolds number leads to an increase

of both the total-to-total- pressure ratio and efficiency. This is explained by the stabilization effect on the boundary layer by the increase of the Reynolds number.

The performance of the rotor alone shows very clearly the hub loaded characteristics of this compressor stage. Following the approach of Greitzer et al. [43], the diffusion factor and the non-dimensionalized pressure rise trough the blade row have been determined. The evolution of these two parameters shows that the diffusion factor will first reach the limit of 0.6. This indicates that the compressor will present a blade stall and the absence of influence of a casing treatment on the stability limit of the compressor.

The influence of a 3x1 mm circumferential groove casing treatment located at 15% of the axial chord downstream the leading edge has been investigated. The effect of the casing treatment from choke to design operating point is very low. If an increase of the pressure ratio can be observed, it stays close to the uncertainty range. As to the efficiency, this change stays far under this range and is thus negligible. At near stall operating point, an increase of 3% of the mass flow, 1% of the pressure ratio and 2% of the efficiency has been observed. These variations correspond to a reduction of the blockage in the tip region and not to an increase of the stall margin. This is confirmed by the measurement of the mid-span pressure ratio over the whole operating range and for various rotational speeds from 40% to 105% of the nominal speed. Indeed, if the casing treatment introduces a slight pressure ratio improvement, it does not influence the stability limit for any rotational speed.

The comparison of the results from the measurements on the compressor stage and the simulations shows a similar trends. Nevertheless, the CFD simulations do not allow reaching a reduced mass flow as low as in the experiments. This is due to the importance of the unsteady phenomena driving the flow at these conditions, which are difficult to capture by steady RANS computations. In consequence, the near stall operating point cannot be simulated, and this for all CFD codes and turbulence models. The simulations performed with the elsA code with various turbulence models show very high discrepancies in terms of computed global performance. This highlights the influence of the turbulence model on the results and points to the fact that great care is required when decision are taken based on CFD computations. All the computations over-estimate the total-to-total pressure ratio and under-estimate the efficiency. This can be explained by differences between the ideal geometry considered by the CFD and the real geometry resulting from the manufacturing and the assembly, by the use of an unadapted turbulence model, and by the flow parameters averaging trough the mixing plane in the computations. This third point is confirmed by the very good agreement between the experimental and numerical performance of the rotor alone. The two CFD codes running with the same turbulence model present opposite conclusion about the effect of the circumferential groove casing treatment in terms of both total-to-total pressure ratio and efficiency. Furthermore, the impact of the casing treatment on the performance is lower than the influence of the turbulence model used for the computation with an identical CFD code. This again shows the low

level of confidence that we can have in numerical simulations for this application.

As mentioned above, the numerical simulations of the compressor stage present high discrepancies in the results. This can be related to various parameters: the turbulence model, the geometry considered, the mesh considered, and the code used. The influence of the turbulence model has been investigated by simulating the compressor with the same mesh and CFD code (elsA by Cenaero) and with three turbulence models. This influence was found important. The exact influences of the other parameters are not highlighted by the work presented here. Indeed, the compressor was simulated with two CFD codes but with also different mesh and different version of the $k - \epsilon$ turbulence model. The effects of these three parameters are mixed in the final result. Additional parametric study on these various elements has to be performed in order to determine the importance of the impact of each parameter.

The experimental and numerical overall performance of the compressor stage for the various configurations investigated are summarized in table 4.4.

			Point A	Point B	Point C	Point D
Exp.	Cruise	<i>Qmred</i>	0.086	0.084	0.0769	0.0532
		PR	1.1007	1.1178	1.1268	1.1012
		Eff.	0.6772	0.7502	0.8085	0.7385
	Take-off	<i>Qmred</i>	0.0871	0.0855	0.0757	0.0549
		PR	1.1013	1.1243	1.1335	1.1056
		Eff.	0.6641	0.7601	0.8528	0.7632
	Casing Treat.	<i>Qmred</i>	0.0861	0.0842	0.0772	0.0544
		PR	1.0990	1.1189	1.1273	1.1035
		Eff.	0.6855	0.7688	0.8083	0.7606
elsA	k- ϵ Smooth casing	<i>Qmred</i>	0.0860	0.0845	0.0765	0.0585
		PR	1.1076	1.1201	1.1301	1.1162
		Eff.	0.6421	0.7042	0.7765	0.7684
	k-l	<i>Qmred</i>	0.0858	0.0841	0.0761	0.0585
		PR	1.1030	1.1143	1.1254	1.1168
		Eff.	0.6077	0.6693	0.7497	0.7559
	k- ω	<i>Qmred</i>	0.0864	0.0853	0.0774	0.0620
		PR	1.1071	1.1214	1.1318	1.1247
		Eff.	0.6442	0.7141	0.7894	0.8116
	k- ϵ Casing Treat.	<i>Qmred</i>	0.0860	0.0846	0.0765	0.0596
		PR	1.1125	1.1245	1.1324	1.1194
		Eff.	0.6597	0.7189	0.7834	0.7781
FINE	k- ϵ Smooth casing	<i>Qmred</i>	0.0856	0.0842	0.0772	0.0568
		PR	1.1111	1.1257	1.1330	1.1042
		Eff.	0.6577	0.7310	0.7893	0.7237
	k- ϵ Casing Treat.	<i>Qmred</i>	0.0861	0.0844	0.0783	0.0580
		PR	1.1081	1.1241	1.1309	1.1085
		Eff.	0.6413	0.7221	0.7781	0.7408

Table 4.4.: Summary of the experimental and numerical global performance of the compressor stage at two Reynolds numbers for the smooth casing and casing treatment configurations

Chapter 5.

Rotor Unsteady Flow Field

As presented in the introduction of this thesis, the three-dimensional unsteady phenomena have a strong influence on the global performance of the low aspect-ratio and highly loaded stage compressor investigated in this work. Furthermore, these phenomena are at the origin of the unstable regime reached at low flow coefficient. If the relation between these phenomena and the instability limit of a compressor has been demonstrated by numerous authors, the exact process of instability inception is not totally understood. Especially in the case of low-pressure compressors for which the amount of references is very limited.

This chapter presents the experimental investigation of the unsteady flow field in the VKI R4 single stage low-pressure axial compressor. This investigation is based on the measurements of the unsteady wall casing static pressure and the total unsteady pressure downstream of the rotor. The experimental results are compared with the numerical simulations in order to firstly validate the CFD code, and subsequently to use the numerical flow visualizations to explain the unsteady phenomena occurring in the compressor stage.

The detailed survey of the tip flow characteristics and its related vortex has been performed by means of fast response pressure transducers located in the rotor casing. This survey has been carried out under choke and design conditions, and near the stability limit. The influence of the Reynolds number on the unsteady flow field has been studied, based on the comparison of the pressure measurements under Reynolds numbers representative of cruise and take-off conditions. Furthermore, the impact of a single circumferential groove casing treatment on the tip leakage vortex has been investigated as well.

The tip leakage vortex path and its extension downstream of the rotor have been determined by simultaneous acquisition of the casing wall static pressures and the total pressure in the main flow. The measurement of the total pressure downstream of the rotor was performed with a miniature fast response single sensor probe. Moreover, the establishment of the unsteady total pressure map downstream of the rotor allows the observation of the three-dimensional hub-corner separation, which becomes a hub-corner stall with a decreasing flow coefficient.

5.1. Unsteady Pressure Instrumentation

5.1.1. Casing Wall Pressure Sensors

The study of the unsteady phenomena occurring in the tip region of the rotor blades is performed by the use of high frequency response instrumentation. The blade-passing frequency being of the order of 6.3 kHz, the sensor must have a bandwidth going well above this value in order to catch the expected phenomena described by its fundamental frequency and its harmonics. The pressure transducer Validyne, used for the determination of the the global performance of the compressor in chapter 4, has a maximum frequency response of around $10Hz$, *i.e.* much too low to see the blade passing in front of it. The use of high frequency response pressure transducers is hence necessary. The sensors chosen are manufactured by the US company Kulite and referenced XCS-062. They will be simply called Kulites hereafter. These are $15PSI$ ($\approx 1bar$) differential pressure sensors with a claimed natural frequency of circa $200kHz$, which is high enough for our purpose since it is more than 30 times higher than the blade-passing frequency. A picture and a drawing of this 1.7 mm external diameter sensor are shown on Fig.5.1 and 5.2.

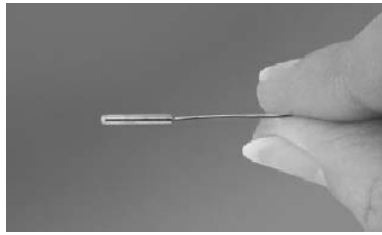


Figure 5.1.: View of the XCS-062 sensor

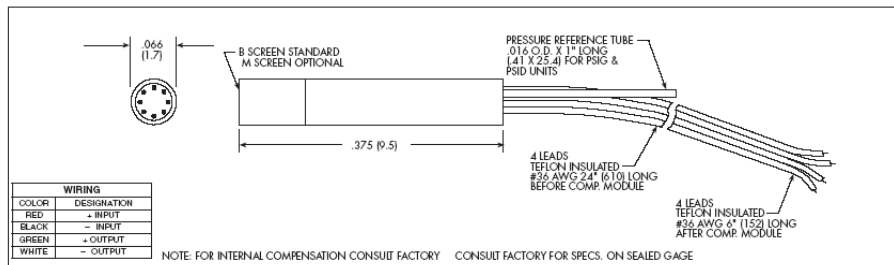


Figure 5.2.: Drawing of the XCS-062 sensor

The principle of this kind of transducer is piezoresistive and is based on the deflection of a membrane due to a pressure difference between its two sides. This membrane separates the measured pressure from a reference pressure. Four strain gauges connected as a Wheatstone bridge are diffused into the silicon membrane. The variation of the pressure difference between the two sides of the membrane induces its deflection. The resistance of the strain gauges varies in function of their elongation/contraction, leading to a variation of the voltage output of the Wheatstone bridge. The main advantages of this kind of transducers come from their very small size. It allows a very high frequency response thanks to the low mechanical inertia of the system. Furthermore the small size gives the opportunity to position the sensing element, the membrane, at the front of the sensor, and thus close to the measurement point. This allows avoiding the presence of a connecting tube ahead of the membrane, which dramatically decreases the frequency response of the system. This type of electronic circuit being dependent on the temperature, a temperature compensation circuit is added to the system.

The fast pressure sensors are placed in the casing of the rotor, allowing a measurement of the unsteady wall static pressure and a visualization of the phenomena associated with the tip leakage flow. The tip gap being only 0.3 mm , the risk of the rotor blades touching the sensors is not negligible. In order to protect these costly sensors, and also to make their handling easier, the Kulites are mounted inside brass inserts that are fixed into the casing of the rotor, Figures 5.3 and 5.4.

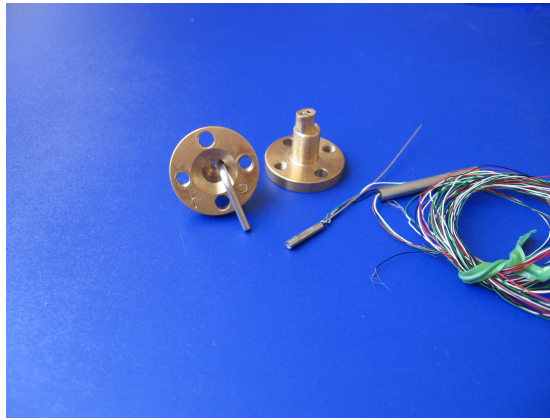


Figure 5.3.: Brass inserts for the Kulites

The use of the inserts allows the establishment of a slight security recess. The schematic view of this configuration is shown on Fig.5.5. The presence of the 0.2 mm depth recess between the membrane and the casing wall ensures that no blade will touch the sensor, which would lead to the destruction of this relatively



Figure 5.4.: Brass inserts including the Kulites mounted inside the casing

costly device. Furthermore, this configuration, with inserts, gives the possibility to reduce the pressure sensing area compared to the blade thickness. The fast response pressure sensors have a diameter of 1.7 mm , corresponding to 120% of the maximum blade thickness at the tip (1.42 mm). However, in order to keep a sufficient bandwidth for the measurement system, the diameter of the sensing hole was kept at 0.8 mm . This size corresponds to 56% of the maximum blade thickness or 4.2% of a blade pitch at tip. The reduction of the natural frequency of the measurement system because of the cavity will be determined by the frequency analysis of the signals and presented in section 5.2.2. The static calibration of the Kulites and pressure taps have been done daily following the same procedure as for the steady pressure measurement.

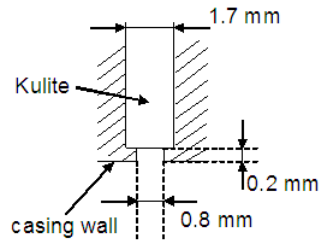


Figure 5.5.: Schematic view of the cavity in front of the Kulite

In every insert, the fast pressure measurement is coupled with a steady pressure measurement by means of a pressure tap connected to a low frequency response pressure transducer from the company Validyne. This steady pressure measurement is used to confirm the absence of drift in the unsteady measurement. This is done by a comparison of the steady pressure with the average of the unsteady

pressure.

5.1.2. Unsteady Total Pressure Probes

Simultaneously with the casing wall static pressure, the unsteady total pressure downstream of the rotor was measured using the miniature fast response pressure probe AP1-C25, Fig.5.6. Full details about the probe design, characteristics and calibration can be found in Brouckaert [11]. This fully cylindrical probe has an external diameter of 2.5 mm and is equipped with a 0.3 mm pneumatic total pressure tap (SR-P01), a $12.5\text{ }\mu\text{m}$ half-shielded chromel-alumel thermocouple (SR-T01), and the unsteady pressure sensor (FR-P01) with a sensing diameter hole of 0.5 mm . This hole provides a recess of 0.4 mm which results in a theoretical frequency response of 127 kHz [11]. The fast pressure sensor has a pressure range of 15 PSI with the vacuum as reference pressure. At $Mach = 0.6$, the probe is insensitive to incidence (within 1%) in an angular range of $\pm 11.8^\circ$. The rotor exit radial traverses were performed in the same measurement plane as the last casing insert (K9), *i.e.* 45% of the axial chord (C_{ax}) downstream the rotor trailing edge, from 16.3% to 97.8% of the blade height. During the radial traverses, the probe was set at the average rotor absolute exit angle for every operating point. Total pressure was always corrected for incidence by using the average exit angle measured by prior pneumatic 3-hole probe traverses.

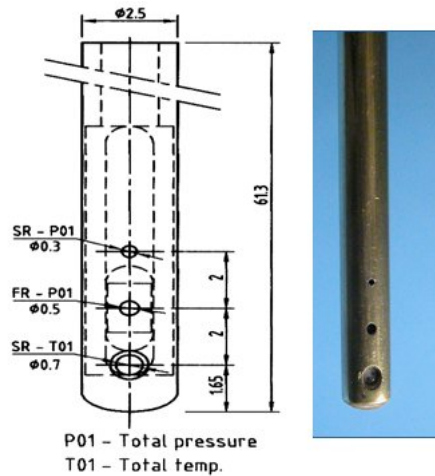


Figure 5.6.: Miniature fast response total pressure probe AP1-C25

The measurement of the unsteady three-dimensional flow characteristics by fast response total pressure probes is still a challenge. The small dimension of the com-

pressor stage under investigation leads to a non-negligible impact of the probe on the measurements. Indeed, the probe AP1-C25, with a diameter of 2.5 mm , represents a flow blockage of 7.95% of the blade pitch (18.85 mm) in terms of frontal area. The effect of the AP1-C25 probe-size on the measurement was investigated by Chomé [20] by means of a comparison with the thinner probe AP1-C15 with an external diameter of 1.5 mm . The spatial extension of the pressure sensing hole (diameter 0.25 mm) represents an averaging over 2.6% of the blade pitch. This means that the probe is unable to detect the contribution of the kinetic energy of all fluctuations with turbulence length scales smaller than the sensing hole dimensions. Regarding blockage, large turbulent length scales are going to be affected by the compression of streamlines around the probe body, resulting in an increase of the axial component and a decrease of the circumferential component of the velocity. Finally, the goal of measuring flow phenomena at the blade tip is complicated by the fact that measurements will be affected by wall proximity effects. All these issues underline once more the need for miniature instrumentation in turbomachines.

5.2. Casing Wall Static Pressure Contour Maps

5.2.1. Wall Fast Static Pressure Sensors Localization

In the base line configuration, *i.e.* for a smooth casing, 9 fast pressure sensors are used for the determination of the wall static pressure field. The disposition of the fast pressure measurement systems is presented on Fig.5.7. They are one rotor pitch apart in the circumferential direction, except for Kulites 8 and 9 that are at the same peripheral position.

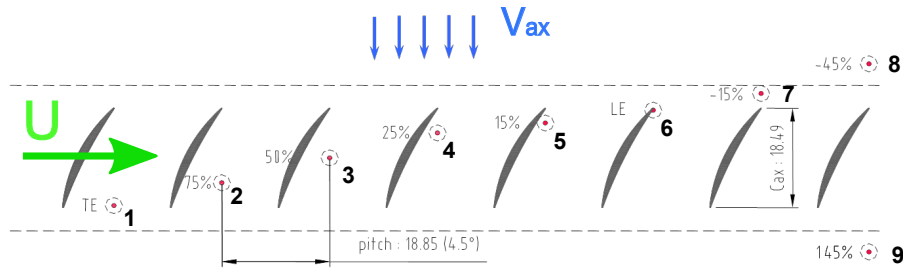


Figure 5.7.: Schematic view of casing static pressure measurement locations

This arrangement allows avoiding a downstream insert being influenced by the wake of an upstream insert. They are distributed along the axial direction from -45% upstream the leading edge of the rotor blade ($K8$) to 45% downstream the trailing edge ($K9$). $K6$ and $K1$ are located at the axial position of respectively the



Figure 5.8.: Photograph of casing with static pressure measurement sensors

leading edge and the trailing edge of the blade tip section. $K5$, $K4$, $K3$ and $K2$ are distributed along the axial direction at 15%, 25%, 50% and 75% of the axial chord. It was expected that the main phenomena would appear in the first part of the blade passage leading to the need of increasing the axial resolution in this region by installing $K7$, $K6$ and $K5$ closer to each other. Furthermore, two supplementary Kulites are placed upstream of the blade row at the same axial position as $K7$ (-15% of the axial chord with respect to the leading edge) respectively at 120° ($K10$) and 220° ($K11$), considering $K7$ at 0° , as shown on Fig.5.9. These three Kulites are used for the detection and the study of the stall phenomena.

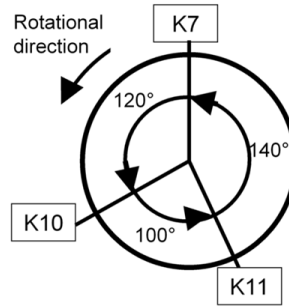


Figure 5.9.: Circumferential arrangement of the sensors used for the stall cell detection

In the casing treatment configuration, the unsteady pressure measurement devices were placed at the same position except the one at $25\% C_{ax}$ downstream the leading edge ($K4$) which was shifted to 30% due to the presence of the groove. Furthermore, Kulite $K5$, situated in the middle of the groove, is shifted radially upwards in order to measure the static pressure at the bottom of the groove. Two additional Kulites were placed at the bottom of the groove, $K1C$ at $9\% C_{ax}$ and $K1D$ at $21\% C_{ax}$, corresponding to the up- and downstream edge of the groove. The bottom of the groove thus contains three static pressure measurement points, two in each corner ($K1C$ and $K1D$) and one in the middle ($K5$). Figure 5.10

presents the fast pressure sensors' arrangement for base line and casing treatment configurations.

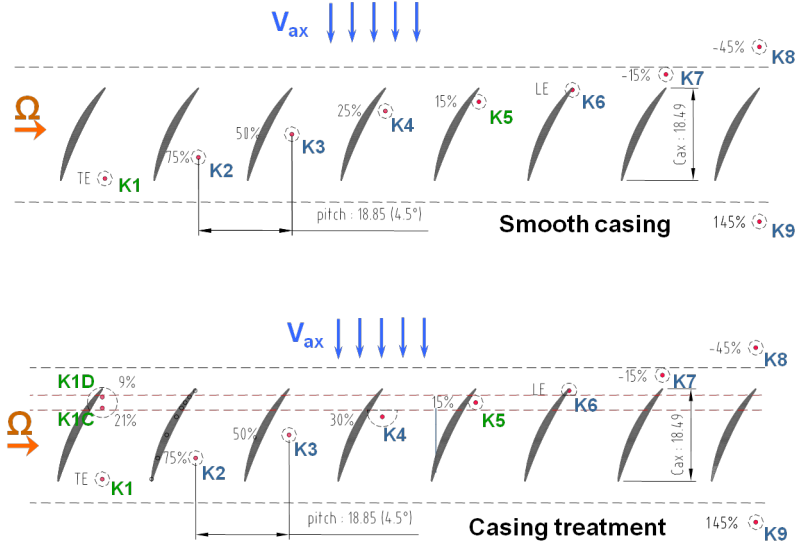


Figure 5.10.: Schematic top view of the casing static pressure measurement locations for both casing configurations

5.2.2. Frequency Analysis

The power spectrum of all the unsteady casing wall static pressure sensors is presented on Fig.5.12. In every trace, the blade-passing frequency, circa 6.3 kHz , appears as a peak with very high energy content. This peak is followed by several harmonics. The energy content at the blade-passing frequency is dependent on the axial position of the sensor. In the trace of the sensor placed over the blade, the peak at 6.3 kHz is very high and, for some of them, more than 20 harmonics are visible. Upstream and downstream the blade row, the energy content is decreasing rapidly. For example, sensor K9, placed at 45% downstream the trailing edge, presents a lower power spectral density for the peak corresponding to the blade-passing frequency, and picks up only 4 to 6 harmonics. This is also confirmed by the power spectrum of the fast response total pressure probe placed in the same axial measurement plane.

The second peak in the power spectrum is related to the natural frequency of the measurement system. This peak is caused by the resonance of the membrane of the sensor itself. The natural frequency of the sensors is between 185 and

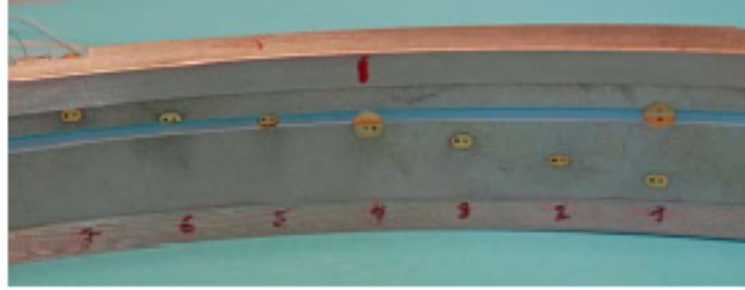


Figure 5.11.: Photograph from the bottom of the rotor casing with the unsteady static pressure measurement systems in the casing treatment configuration

215 kHz . Because of the acquisition frequency used (300 kHz) and the absence of anti-aliasing filter, this natural frequency is visible between 85 and 115 kHz .

This frequency analysis shows that the frequency contents of the fast pressure signals are clear, showing the presence of only the blade-passing frequency with a large number of harmonics, and the natural frequency of the measurement system. All fast response pressure signals are low-pass filtered at 80 kHz for the casing pressure, and at 50 kHz for the rotor exit traverse probe, since no more than 4 to 6 harmonics are picked up at this location.

5.2.3. Post-processing procedure

In order to extract the important information from the raw unsteady pressure signals, various post-processing methods have been applied. The post-processing allows mapping the mean unsteady casing wall pressure and the fluctuations around this mean value. The post-processing methods consist in determining the mean pressure profile over one pitch, by using the Phase-Locked Averaging process, and the interpolation of the pressure between the measurement planes.

The first step of the post-processing is to extract the mean value of the unsteady wall static pressure over one single pitch from the large number of blade passages acquired during each test. The unsteady wall static pressures are acquired during a period of 218.5 ms . For a rotational speed of the rotor row of around 4728 RPM, this corresponds to more than sixteen rotor revolutions, or 1377 blade passages. In order to acquire around 50 points per blade passage, and with a blade-passing frequency of circa 6.3 kHz , the sampling frequency used for the fast pressure sensors was chosen equal to 300 kHz . An example of the raw voltage signal over one revolution given by fast pressure sensor *K3*, placed at mid-chord of the rotor blade, is presented on Fig.5.13 (top). The oscillations of the pressure, visible on this figure, are due to the pressure variation between the pressure- and suction side of the

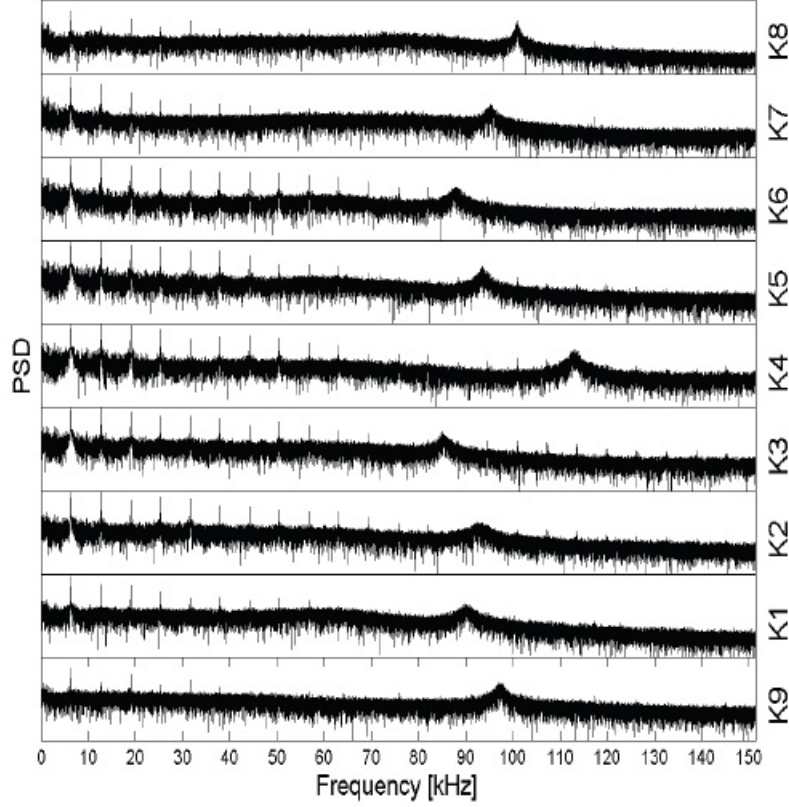


Figure 5.12.: Power spectral density of the unsteady casing wall pressure signals for cruise conditions under design point

blades, and thus correspond to the blade passages. Figure 5.13 (bottom) shows the one-per-revolution trigger signal. This signal is used for the determination of the revolutions of the rotor.

Figure 5.14 presents a zoom on the signal from fast sensor *K3*. On this figure the voltages are converted into pressures relative to the tank, using the linear calibration law. The blade passages can clearly be identified as taking place between two pressure drops, corresponding to the blade passing in front of the sensor. The left and right hand side of the pressure drop correspond respectively to the pressure- and the suction side of the blade. The “strong” periodicity of the unsteady pressure signals, corresponding to the blade-passing frequency, justifies the averaging of these signals over one blade passage. This is called the phase locked averaging procedure and can be explained as follows.

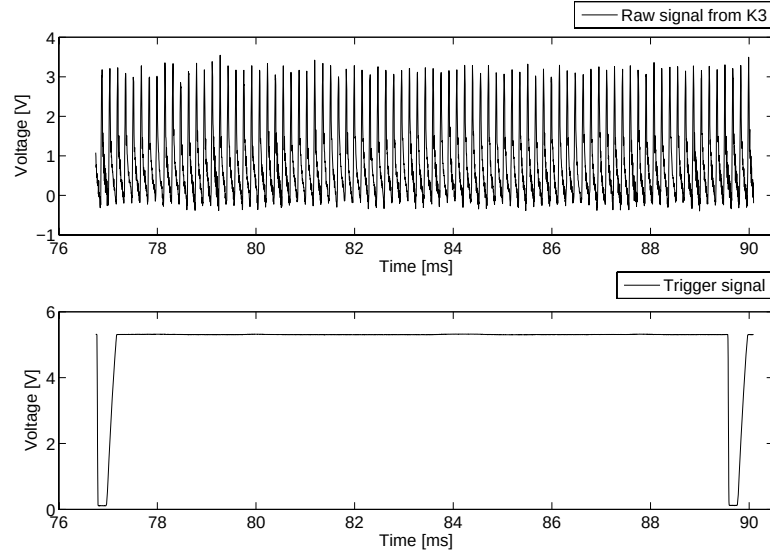


Figure 5.13.: Raw signal from the mid-chord fast pressure sensor K3 (top) and from the trigger (bottom) over one rotor revolution

Between two pulses of the trigger signal, *i.e.* over one single revolution of the rotor, each fast pressure sensor sees 80 blade passages. For every blade passage, a fast sensor acquires approximately 50 pressure measurements. The signal over each complete acquired revolution is interpolated on 8000 points. This corresponds more or less to a multiplication by 2 of the number of points per revolution. The interpolation of the signal at the beginning and at the end of the revolution is performed by using the last point of the previous revolution and the first one of the next revolution. Considering the rotational speed of the rotor to be constant over one single revolution, each blade passage corresponds exactly to 100 points. All the interpolated blade-passage signals are then averaged point by point. The signal over one blade passage obtained by using this method is an average of all the blade passages and is called the Phase Locked Average (*PLA*).

The *PLA* of the mid-chord unsteady pressure signal is presented on Fig.5.15. On this plot the *PLA* is compared to two random blade passages. The three signals are non-dimensionalized by the inlet total pressure P_{01} at midspan, and are plotted in function of the circumferential coordinates, knowing that one pitch is equal to :

$$\frac{360deg}{80blades} = 4.5deg. \quad (5.1)$$

The averaging effect is clearly visible since the fluctuations on the time series

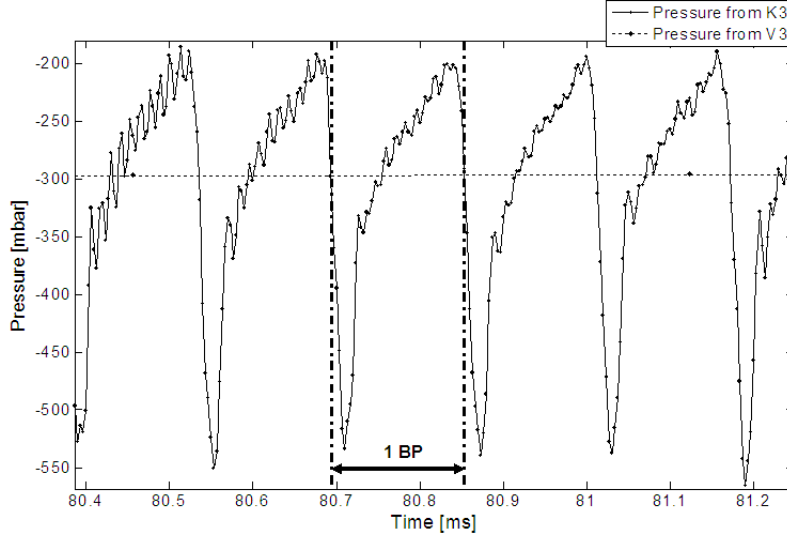


Figure 5.14.: Relative unsteady pressure at mid-chord over 5 blade passages

disappear on the average. This also shows that a RMS value can be computed over one blade passage, showing interesting features. The average is performed over 1280 blade passages, corresponding to 80 blade passages over 16 revolutions. From a statistical point of view this number is significant.

The *PLA* determination method is based on the assumption that the rotational speed of the rotor is constant over one single revolution. The variation of the velocity of the VKI-R4 facility during the acquisitions was investigated by Chomé [20]. He determined that the maximum variation of the peripheral velocity over one single acquisition equals 0.38%. This is based on the one-per-revolution trigger and thus corresponds to the variation between the revolutions. Thanks to the mechanical inertia of the facility, the reduced speed evolution over a single revolution can be considered linear. Hence the maximum velocity variation between the revolutions corresponds to the maximum velocity variation between blade passages over one single revolution. This maximum variation of the blade-passage time of 0.38% is totally negligible.

The phase-locked averaging procedure allows to obtain a first 3D visualization of the unsteady casing wall static pressure field. This is presented on Fig.5.16. If this visualization allows a comparison between different configurations and between experimental and CFD results, it is not enough to catch the flow structure in the tip region of the rotor blades. For that, a spatial interpolation of the pressure over the entire measurement domain is required. The interpolation process is characterized by four principal parameters: the domain of interpolation, the quality of the final

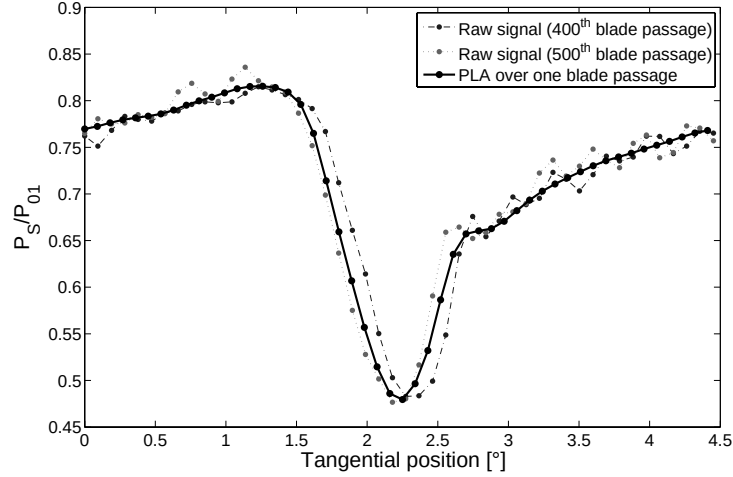


Figure 5.15.: PLA of the mid-chord pressure signal compared to two random blade passages

grid, the method of the interpolation, and the interpolation direction.

The interpolation domain considered is delimited in the axial direction by fast pressure sensors *K8* and *K9* placed at 45% of the axial chord, respectively upstream of the leading edge and downstream of the trailing edge. In the circumferential direction, the interpolation will be carried out over one single pitch, corresponding to the result of the PLA process. For the quality of the final grid, the interpolated pressure will be given on a 200x200 grid. This is a good balance between the high quality grid and the limitation of the processing time. The imposed method for this interpolation is linear.

Concerning the direction of the interpolation, the choice was really complicated by the difference in terms of resolution of the measurement points between the axial and the circumferential direction. The resolution in the axial direction is very low because it is determined by the nine measurement planes, corresponding to the fast sensors' locations. On the other hand, the circumferential resolution is high with 100 points over one pitch from the PLA. The problem of the choice of the interpolation direction was already underlined by Gannon et al. [39]. Due to the resolution difference, the basic interpolation in the axial and circumferential direction gives a jagged pressure profile around the blade and does not allow the observation of the physical phenomena in the flow, like the tip leakage vortex. The static pressure field based on this interpolation is shown on Fig.5.17 for the design point in the smooth casing configuration. A second reason of this shape of the pressure field is the direction of the strong pressure gradient. This pressure

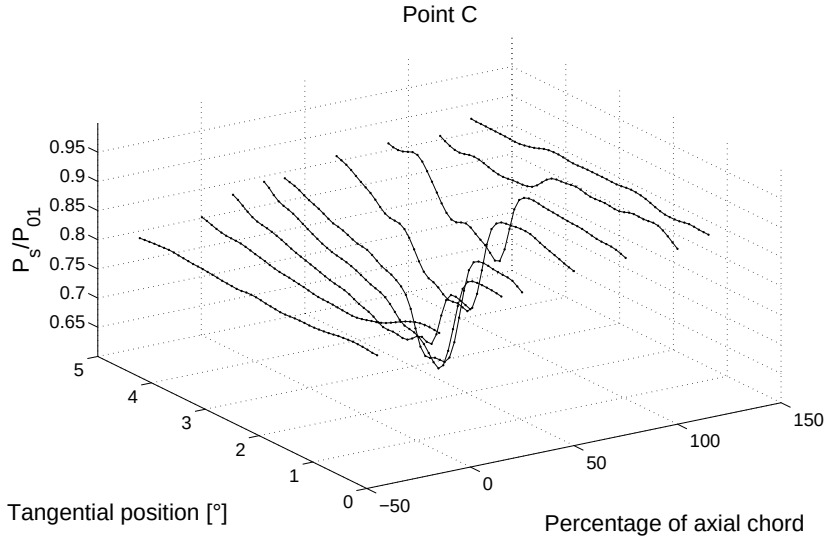


Figure 5.16.: 3D visualization of the phase-locked averaged casing wall static pressure under design conditions

gradient is due to the passage of the blade in front of the fast pressure sensors. Consequently this gradient follows the blade stagger angle and is not aligned with the axial or the circumferential axis.

It was then decided to interpolate the wall static pressure following the stagger angle at the tip. The interpolated pressure field is presented on Fig.5.18 for the smooth casing configuration under design (left) and near-stall conditions (right). This interpolation allows to remove the wave shaped isobars in order to reveal the physical phenomena occurring in the flow.

The first observation is the location of the strong pressure gradient corresponding to the pressure side of the blade. Secondly, the increase of the static pressure in the axial direction over the rotor row is clearly visible at design point. Under near-stall conditions this is less obvious because of the limited pressure ratio achieved by the rotor. Also at design point, a low pressure region along the suction side of the blade is present. This region corresponds to the origin of the tip leakage vortex, as described in the literature review. From this region, a low pressure path spreads into the flow passage. This path is generated by the rolling up process of the tip-leakage flow into the tip leakage vortex. This vortex extends in the flow passage and its axis is aligned with the low pressure path. The importance of the pressure gradient in the tip leakage vortex path decreases in the axial direction. This is due to the radial plunge of the vortex into the flow passage as it moves downstream.

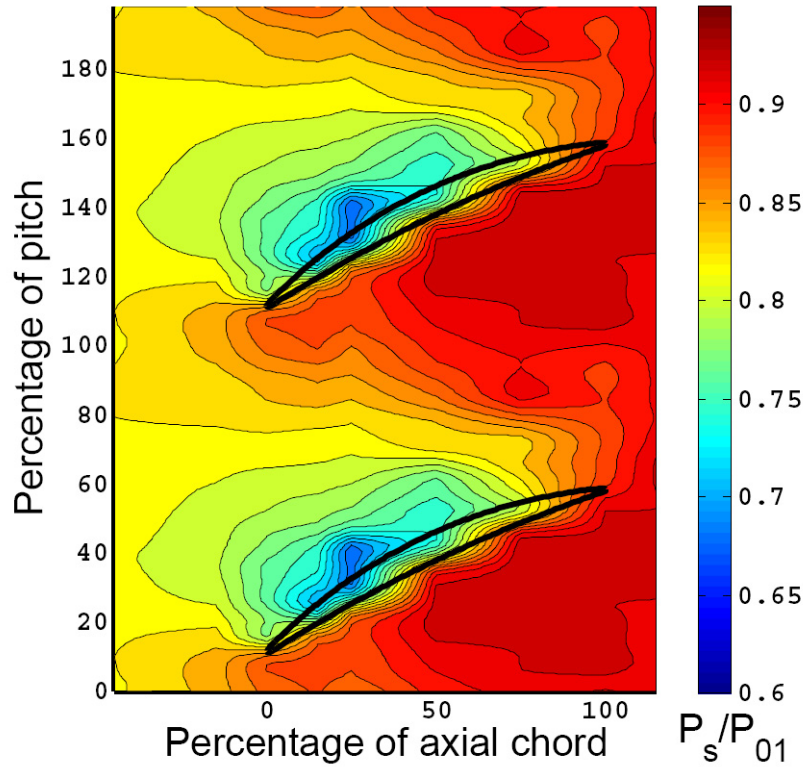


Figure 5.17.: Static pressure field, under design conditions for smooth casing configuration, based on an interpolation along the axial direction

Under near-stall conditions, presented on the right hand side of Fig.5.18, the low pressure region, corresponding to the origin of the tip leakage vortex, is present but not clearly delimited. Furthermore, the extension of the tip leakage vortex, indicated by the low pressure path spreading in the flow passage, cannot be located with accuracy. This is due to the big difference between the stagger angle of the blade, considered as the interpolation direction, and the angle of the vortex axis. Indeed these two directions follow the two pressure gradients present in the flow. In the same way that the interpolation along the axis is not appropriate for a pressure gradient following the stagger angle, an interpolation along the stagger angle is not relevant for a pressure gradient following a very different direction. In the design case, the stagger angle and the vortex axis angle are close. The interpolation along the stagger angle can be applied for the tip leakage vortex. It

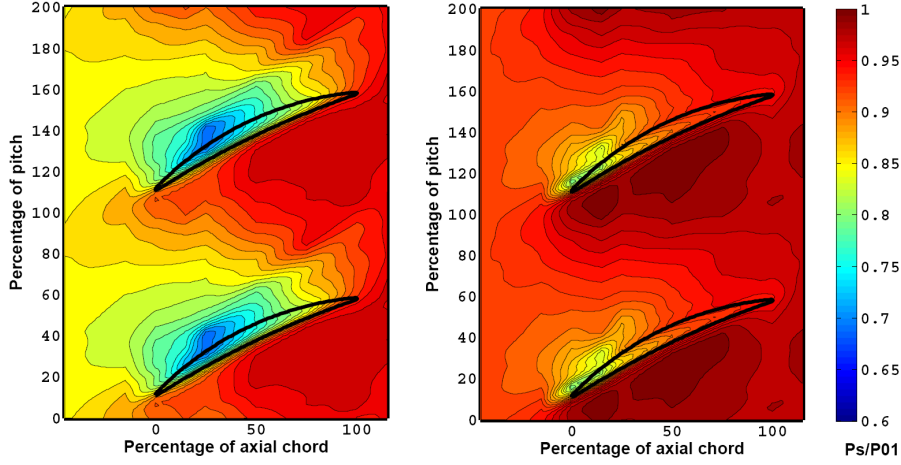


Figure 5.18.: Static pressure field, under design (left) and near-stall (right) conditions for smooth casing configuration, based on an interpolation along the stagger angle direction

is no longer the case under the near-stall conditions. A new set of directions of interpolation, depending on the working conditions and on the spatial location, has thus to be defined. These directions have to be as close as possible to the physical phenomena expected in the flow. This process was suggested by Gannon et al. [39]. Finally, the interpolation has been performed, for all working conditions except near stall, following the axial direction upstream the leading edge and downstream the trailing edge, and following the vortex direction between the leading- and the trailing edge. The choice of the vortex direction instead of the stagger angle comes from the fact that we are interested in the tip leakage vortex phenomenon. Under near-stall conditions, the interpolation direction is also axial upstream the leading edge and downstream the trailing edge. Between mid-chord and the trailing edge, the pressure field is interpolated along the stagger angle. Between the leading edge and the mid-chord, one carries out an interpolation between two directions: one following the stagger angle, and the second one following the tip leakage vortex axis. A multiple linear regression is performed between these two directions.

The direction of the vortex axis, used for the interpolation under near-stall conditions, as well as the regions where different directions of interpolation are applied, are determined graphically based on the results obtained from the interpolation following the stagger angle. This process is long and must be adapted for every compressor or working conditions.

5.2.4. Tip Leakage Vortex Path

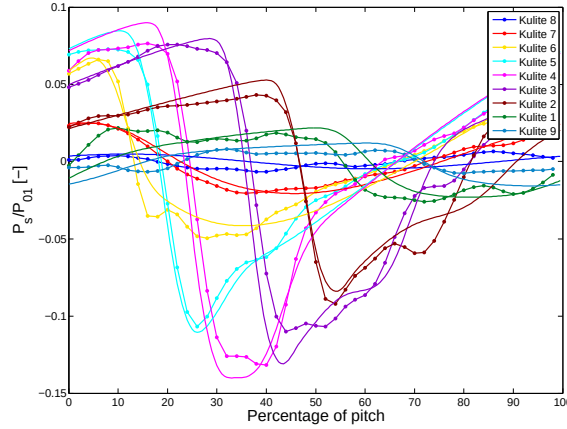


Figure 5.19.: Experimental (dot) and averaged numerical (single line, FINE by B. Farkas) non-dimensionalized unsteady casing static pressure variation at design working point at various axial positions under cruise conditions

The unsteady wall casing static pressure distribution under design conditions over one pitch is presented on Fig.5.20. This distribution is obtained by applying the phase-locked averaging method presented above. The traces correspond to the variation of the static pressure around its mean value for the different fast sensors, non-dimensionalized by the mid-span inlet total pressure $\frac{P_s - P_{smean}}{P_{01}}$. On this figure, the experimental distribution is compared to the numerical results.

The pressure traces show two different regions. The first region comprises a sharp pressure drop followed by a very low pressure area. This pressure drop occurs when the pressure side of the blade is passing by the pressure sensing hole. The location of this pressure drop varies in function of the axial position of the fast pressure sensor depending on the stagger angle. The pressure drop is followed by a low pressure area corresponding to the blade passage in front of the sensor and caused by the very high velocity of the flow inside the tip clearance, as described in the literature review. The pressure trace from the sensor located at mid-chord (sensor *K3*) shows that this low pressure region extends to about 7.5% of the rotor pitch, which corresponds to the maximum thickness of the blade at this position. The end of this area corresponds to the passage of the suction side of the blade by the sensing hole.

The second region is characterized by an increase of the pressure from the suction side to the pressure side of the blade and corresponds to the flow passage. In some

traces, a second local pressure minimum is visible. This is attributed to the tip leakage vortex itself rolling-up into the passage.

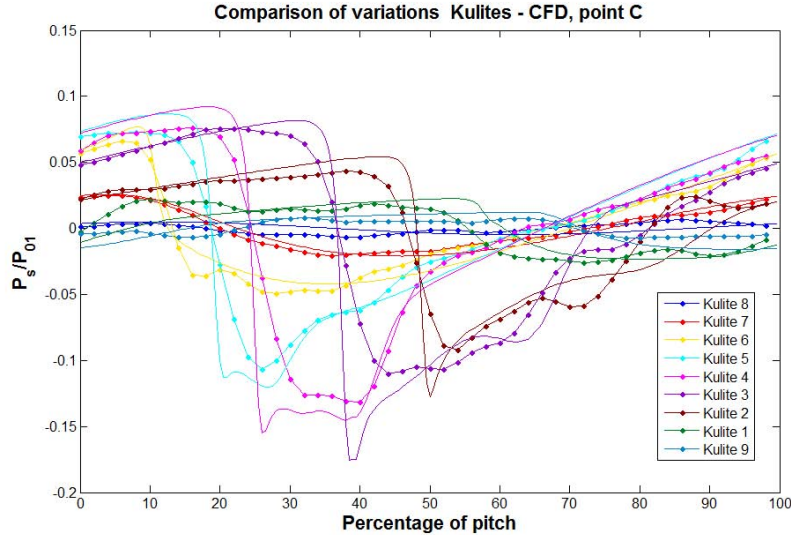


Figure 5.20.: Experimental (dot) and numerical (single line, FINE by B. Farkas) non-dimensionalized unsteady casing static pressure variation at design working point at various axial positions

The agreement between CFD and experiments can be considered very satisfactory in terms of pressure drop location and pressure levels. Differences can be observed in the sharpness of the pressure drop as well as in the absence of some sharp pressure peaks in the results of the experiment. This is caused by the spatial extension of the sensing area of the pressure measurement systems. The diameter of 0.8 mm of the sensing holes, which corresponds to 4.2% of the rotor pitch, introduces an averaging of the pressure leading to a smoothing of the pressure traces. The spatial averaging of the experimental data due to the size of the probe is shown on Fig.5.19. This plot presents the PLA of the experimental casing static pressure traces and the averaged static pressure from the numerical simulations performed by B. Farkas with the FINE code. The averaging of the numerical traces is performed over 10 points corresponding to 5% of the blade passage, and to the size of the sensing area of the static pressure sensors in the inserts. On this figure, we can observe the very high similarity between the experimental and numerical results, especially in terms of slope in the blade region and of amplitude of the traces.

The influence of the Reynolds number on the casing static pressure distribution

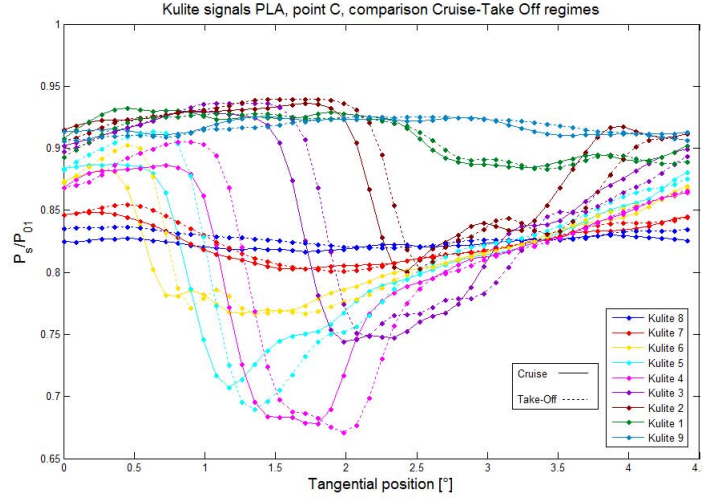


Figure 5.21.: Experimental non-dimensionalized unsteady casing static pressure variation at design working point at various axial positions under cruise and take-off pressure conditions

has been investigated by using the possibility to pressurize the facility, as explained in section 3.5. Figure 5.21 presents the pressure traces from all the fast sensors for conditions corresponding to cruise and take-off at design working point. In section 4.3, it has been shown that the flow characteristics in terms of static-to-total pressure ratio in the compressor at the various working points between cruise and take-off conditions were similar. A comparison between these two working conditions is thus relevant. As it can be observed on Fig.5.21, the effect of the Reynolds number on the pressure level and the shape of the pressure distributions is very low. But, on the other hand, a phase shift appears between the results under cruise and take-off conditions. This shift is caused by the difference of the torque required to run the compressor under low- or high pressure, and caused by the torsion of the shaft between the rotor and the measurement location of the one-per-revolution trigger. Under low-pressure conditions, corresponding to cruise conditions, the torque required to run the compressor (180 Nm) is really lower than under high pressure, corresponding to take-off conditions (450 Nm). This torque difference introduces a variation in the torsion of the shaft. This and the distance on the shaft between the rotor and the trigger location lead to a change in the relative position of these two elements. It explains the phase shift in the casing pressure traces between the two test conditions. Consequently hereafter, the take-off data are artificially shifted in order to consider the same circumferential

reference relative to the rotor. The very low influence of the Reynolds number as well as the phase shift were also observed at the other working points (points *A*, *B* and *D* as described in section 3.5).

The effect of the circumferential groove casing treatment on the wall casing static pressure distribution has also been investigated. The study of the pressure traces does not show a clear effect of casing treatment on the flow path. The interpolation of the pressure described in the previous section allows showing this influence.

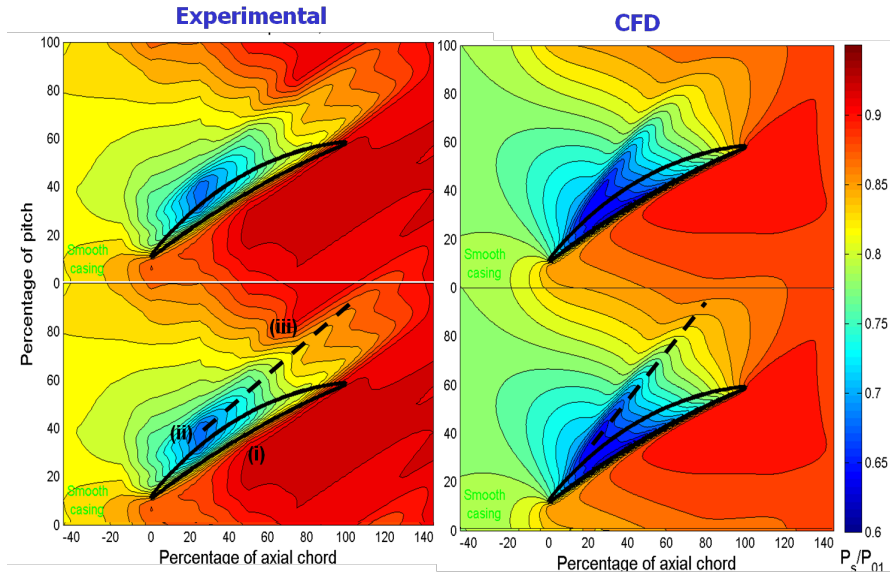


Figure 5.22.: Non-dimensional unsteady casing wall static pressure field from the interpolation of the experimental measurement (left) and from the CFD calculation using FINE by B. Farkas (right) in the smooth casing configuration under design working conditions

The interpolated experimental pressure distributions for the smooth casing configuration at design and near-stall working points are presented on the left hand side of Fig.5.22 and Fig.5.23 (the results for the choke conditions are presented in Appendix B). On these figures the pressure field is presented over two pitches in order to increase the visibility of the phenomena occurring in the flow. The pressure is given in terms of wall static pressure divided by the inlet stage mid-span total pressure P_s/P_{01} and an identical pressure scale is used for both experimental and CFD results. On Fig.5.22, three regions are marked. Region (I) indicates the sharp pressure gradient corresponding to the pressure side of the blade. Region (II) corresponds to the low static pressure region on the suction side of the blade.

As already indicated in the literature survey, this low pressure region represents the origin of the tip leakage vortex (Storer and Cumpsty [102]). From region (II), a region of bended isobars spreads into the flow passage. This pressure trough, indicated as region (III) and marked by the dashed line, is caused by the extension of the vortex into the flow passage. Inoue et al. [58] showed that the tip leakage vortex axis is aligned with this pressure trough. The comparison of the interpolated experimental measurements and the results of the numerical simulation shows that these three regions are clearly visible in the results of the numerical simulation as well. The location of the origin of the tip leakage vortex represented by the low pressure region is well predicted by the CFD (at around 25% of the axial chord under design working conditions). Yet, the direction of the tip leakage vortex trajectory presents a steeper slope in the numerical results. This can be related to the difference between the real tip gap in the compressor stage, and the theoretical tip gap considered in the simulations. In the CFD simulations, the tip gap is fixed at 0.3 mm (1.3% of blade height). But as presented in section 3.2, the real minimum tip gap under running conditions is between 0.12 and 0.25 mm (between 0.5% and 1.08% of blade height), depending on the circumferential location.

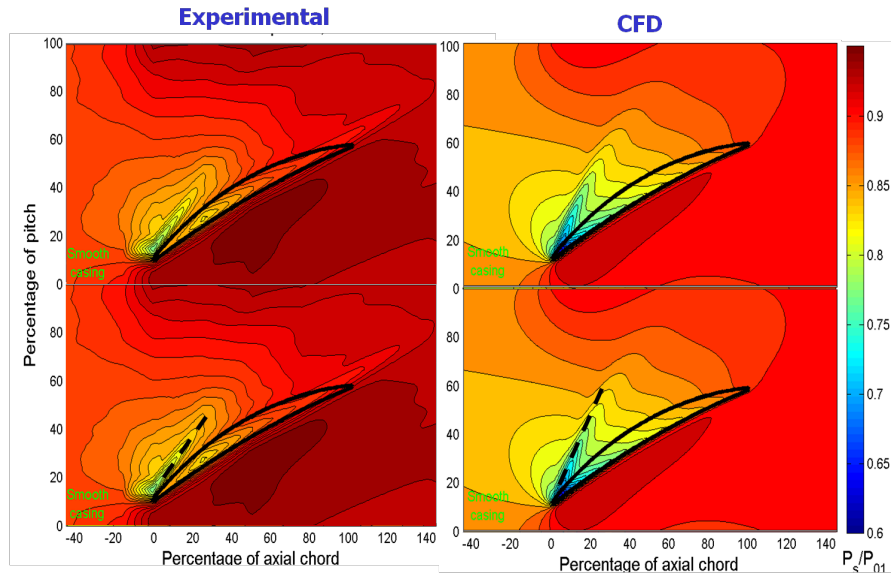


Figure 5.23.: Non-dimensional unsteady casing wall static pressure field from the interpolation of the experimental measurement (left) and from the CFD calculation using FINE by B. Farkas (right) in the smooth casing configuration under near-stall working conditions

The casing wall static pressure field from the interpolation of the experimental measurement and from the CFD calculation, in the smooth casing configuration under near-stall working conditions, is presented on Fig.5.23. The study of this figure and of the previous reveals the influence of the throttling on the origin of the tip leakage vortex and on its slope. A decrease of the flow coefficient leads to a shift of the origin of the vortex, corresponding to the low pressure region, towards the leading edge of the blade. This is caused by the increase of the incidence, and hence of the blade loading, leading to the upstream displacement of the maximum pressure difference between the pressure- and suction side of the blade. Near stall, this region is located nearly at the leading edge of the blade. At the same time, the slope of the vortex axis relative to the stagger angle increases, as observed experimentally by Koch in 1974 [63]. This behavior is confirmed by the unsteady pressure field under choke working conditions (see Appendix B) [12]. The effect of the throttling on the tip leakage vortex comes from the fact that the direction of the vortex results from an equilibrium between the momentum of the main flow and of the tip leakage flow [78]. The reduction of the flow coefficient leads to an increase of the blade loading, *i.e.* of the momentum of the tip leakage flow. At the same time, this reduction induces a decrease of the momentum of the main flow. The equilibrium between the two momentums is thus modified, leading to an increase of the slope of the tip leakage vortex compared to the stagger angle. The increase of the slope of the tip leakage vortex axis could lead to an increase of the interaction region between the vortex and the main flow. It induces the creation of more blockage due to the vortex. This will be discussed in the following section by the observation of the unsteady total pressure distribution downstream the rotor row. Furthermore, the decrease of the flow coefficient leads to a reduction of the visible length of the tip leakage vortex path, represented by the pressure trough. At design point, the vortex is still visible in the wall plane downstream the trailing edge of the blade. At near-stall, it vanishes rapidly from the mid-chord.

In spite of the increase of the slope of the tip leakage vortex, its trajectory is going well into the blade passage and at the same time remaining quite far from being tangentially aligned with the plane of the leading edge of the rotor. Furthermore, the study of the casing wall static pressure field under near-stall conditions does not show any sign of the presence of a backflow from tip clearance fluid at the blade trailing edge, or a leading edge spillage of tip clearance flow below the blade tip of the adjacent blade. According to Tan [106], this absence of total blockage of the core flow in the tip region under near-stall conditions indicates that the compressor presents a typical modal stall inception process linked to a long wavelength disturbance, extending radially through the stage. This type of stall inception of the compressor was already pointed out in section 4.3 by the fact that the stability limit was reached at the zero-slope point of the total-to-static characteristic, indicating a modal type of stall inception.

The comparison between the experimental and CFD results shows also a good prediction of the casing flow field, except for the slope of the tip leakage vortex axis that is again steeper in the numerical results. The numerical simulations under

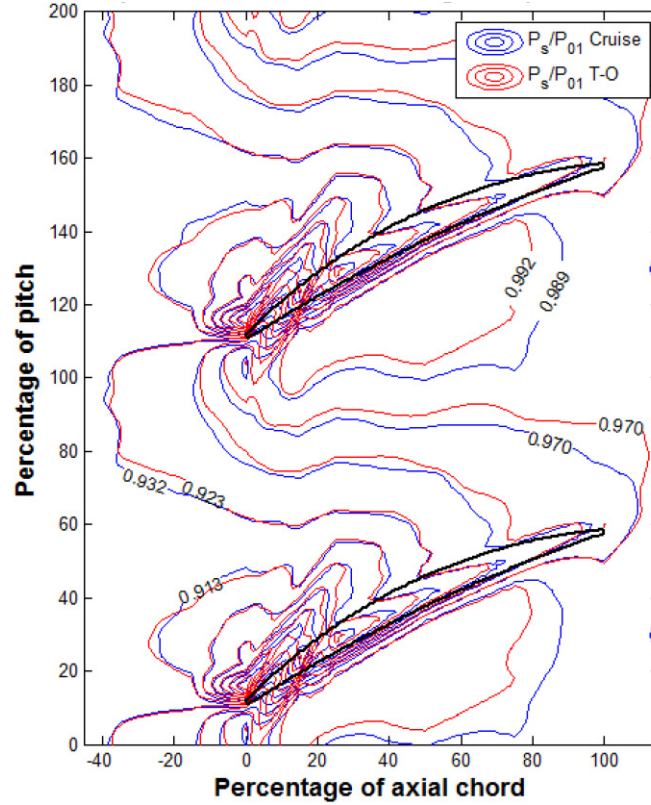


Figure 5.24.: Non-dimensional experimental interpolated unsteady casing wall static pressure difference between the cruise and the takeoff conditions in the smooth casing configuration at near-stall working point

near-stall conditions allow to confirm the absence of any backflow or spillage, as described in the previous paragraph.

The influence of the Reynolds number on the casing wall static pressure field is presented on Fig.5.24. This figure shows the isobar lines of the static pressure, measured with inlet stage total pressure conditions corresponding to cruise and corresponding to take-off, at near-stall working point for both conditions. The isobars correspond to the static pressure, non-dimensionalized by the mid-span inlet total pressure $\frac{P_s}{P_{01}}$. The high similarity between the two pressure fields leads to the assessment that the Reynolds number has no effect on the casing wall static pressure distribution. This weak effect has been observed at the other working points as well.

The experimental casing wall static pressure maps at design working point are

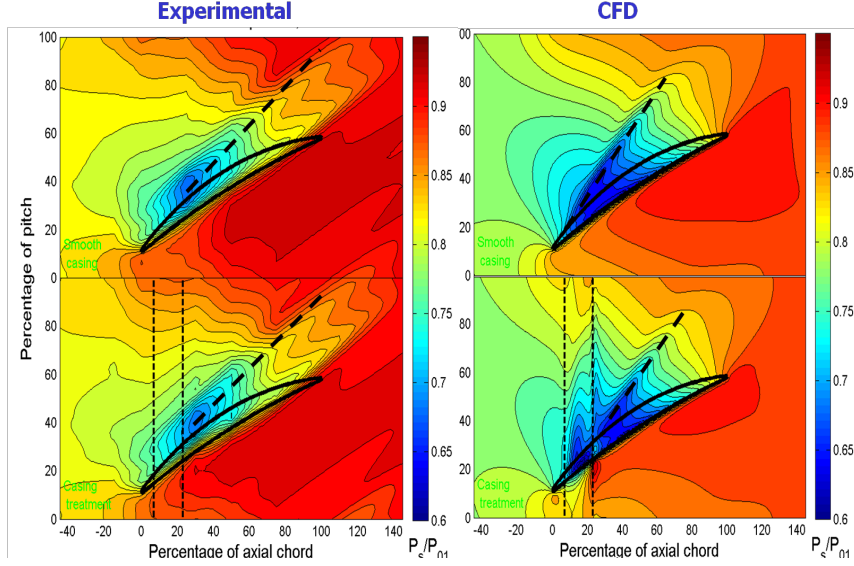


Figure 5.25.: Non-dimensional experimental (left) and numerical (right, FINE by B. Farkas) casing static pressure field for smooth casing (top of the figures) and casing treatment configuration (bottom of the figures) under design working conditions (point C')

compared to the numerical results in Fig.5.25 for both smooth casing and casing treatment configuration. The location of the circumferential groove casing treatment (between 9% and 21% of the axial chord) is represented by black dashed lines. The pressure field at the groove location corresponds to the pressure in the radial plane of the casing, *i.e.* the pressure at the entrance of the groove. In the experimental results, the pressure fields are similar in terms of shape or level of the isobars, as well as of the location of the low pressure region or pressure trough corresponding to the tip leakage vortex. This means that the casing treatment does not influence the wall static pressure field at this working point. On the other hand, the numerical results show an important influence of the circumferential groove on the pressure distribution around the casing treatment. The variation of the static pressure at the groove entrance is obviously not visible in the experimental results, which is due to the impossibility to place fast pressure sensors at this location. Outside the groove area, the impact of the groove casing treatment on the tip leakage vortex path and on the flow is negligible, as observed in the experimental results. These observations made at design working point are valid for choke working point (point A) and point B as well.

Fig.5.26 presents the experimental and numerical static pressure field under

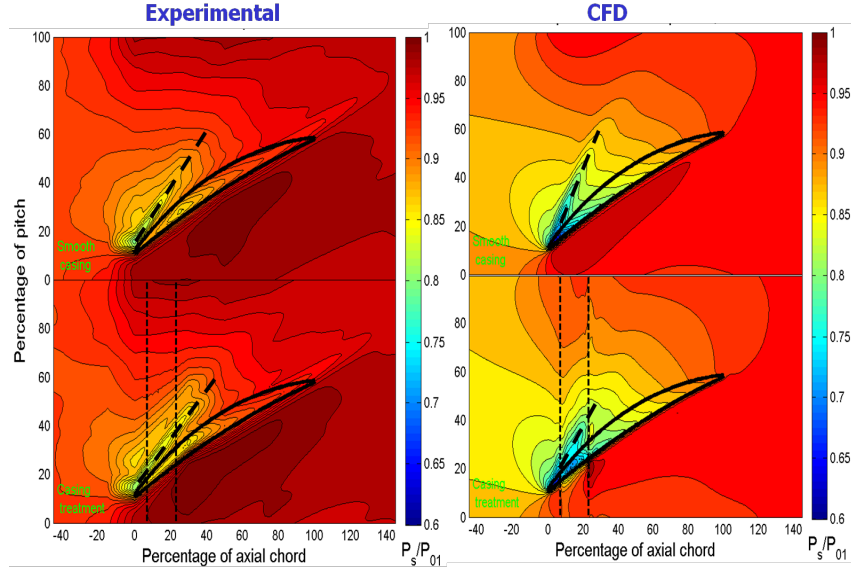


Figure 5.26.: Non-dimensional experimental (left) and numerical (right, FINE by B. Farkas) casing static pressure field for smooth casing (top of the figures) and casing treatment configuration (bottom of the figures) under near-stall working conditions (point D)

near-stall conditions for both casing configurations. Under these working conditions, the circumferential groove has a visible effect on the tip leakage vortex path. The first influence is the reduction of the vortex angle with respect to the axial direction by the use of the casing treatment. This impact is visible in the experimental measurements as well as in the numerical simulations. The second effect is the variation of the extension of the tip leakage vortex in the wall plane. In the numerical results, the use of the casing treatment leads to a decrease of the length of the vortex that can be explained by a radial shift of the vortex towards the hub. In the experimental flow field, the vortex seems to be elongated by the use of the casing treatment. This elongation is explained by the axial displacement of the sensor located just downstream the groove, from 25% to 30% of the axial chord, as described in section 5.2.1. This pressure field variation is thus not caused by a physical phenomenon. Outside the sphere of influence of the tip leakage vortex, the flow field is similar for the two casing configurations.

The impact of the circumferential groove casing treatment on the flow field under the various working conditions can be explained by the position of the origin of the tip leakage vortex relative to the casing treatment. The origin of the vortex, corresponding to the low pressure region, is located downstream the

casing treatment for choke to design working conditions. At these working points, the effect of the groove on the tip leakage vortex is negligible. On the other hand, under near-stall conditions, the effect of the casing treatment on the tip vortex is important. At this working point, the low pressure region is located upstream the casing treatment.

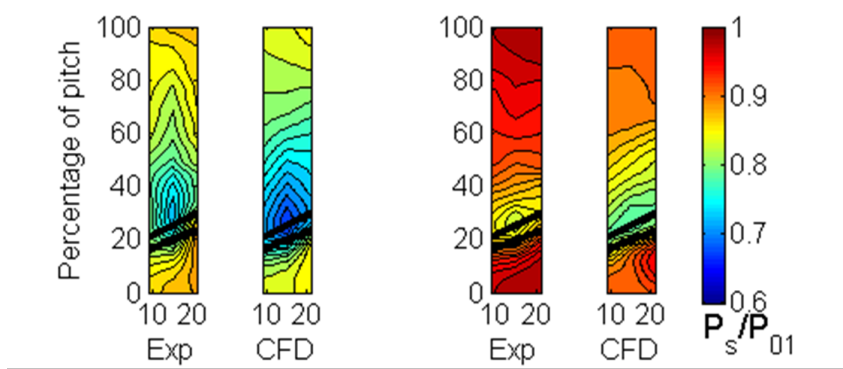


Figure 5.27.: Non-dimensional experimental and numerical (FINE by B. Farkas) static pressure at the top of the circumferential groove casing treatment under design working point (left) and near-stall working conditions (right)

The experimental and numerical static pressure fluctuations occurring at the bottom of the groove are presented on Fig.5.27. The experimental pressure fields are obtained by the interpolation of the pressure traces from the three fast pressure sensors installed at the bottom of the groove. As described in section 5.2.1, two of these sensors are placed on the edge of the groove ($K1D$ and $K1C$ at respectively 9% and 21% of the axial chord) and the third at the center of the groove ($K5$ at 15% of the axial chord). The interpolation is done following the same direction as the one used for the interpolation of the pressure in the casing plane, at the same axial position. The numerical pressure map is the result of an identical interpolation of the three pressure traces extracted from the CFD results at the same axial position as the fast sensors. The pressure fluctuations under design (left hand side on the figure) and near-stall (right hand side on the figure) conditions show a typical sinusoidal shape at the scale of the blade-passing frequency.

The tip leakage vortex can be located by the pressure trough in the casing wall static pressure maps, as described before. This vortex is also characterized by a very high level of turbulence. This high level is caused by the high gradient of the magnitude and the direction of the velocity inside the vortex, as well as by its oscillation. The root mean square (RMS) of the static pressure in the casing plane is thus an appropriate way of determining the tip leakage vortex path. The RMS

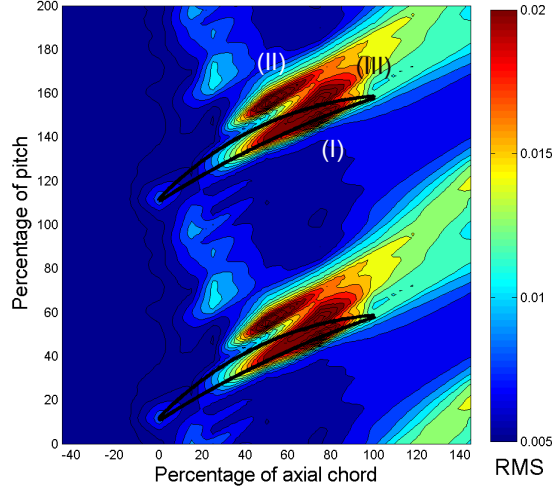


Figure 5.28.: Experimental non-dimensional casing wall static pressure fluctuations around the PLA (RMS) under choke working conditions for the smooth casing configuration

is a statistical tool giving the magnitude of the variation of a quantity around its mean value. In this case, the RMS of the static pressure is based on the variation of the pressure traces of all the blade passages around the trace resulting from the PLA. Consequently, the RMS of the static pressure in one point is defined as:

$$RMS_P = \left(\frac{1}{N-1} \sum_{i=1}^N (P_i - P_{PLA})^2 \right)^{0.5}. \quad (5.2)$$

where N is the number of blade passages acquired, P_i is the pressure at the blade passage i at the point considered, and P_{PLA} is the pressure of the PLA at this same point. From the pressure traces coming from the fast casing pressure sensors and using this definition, it is possible to calculate the traces of the RMS of the pressure at the axial positions corresponding to the various sensors. By means of the same interpolation procedure as the one used for the pressure, the map of the RMS of the casing wall static pressure can be established. Figure 5.28 presents the RMS of the pressure under choke working conditions for the smooth casing configuration. The regions of high level of turbulence correspond to the high RMS regions. There are three noticeable areas of high RMS. The first region (I) is located along the pressure side of the blade. The high RMS at this location is caused by the very high pressure gradient, the oscillation of the blades and the discrete character of the

measurement. The blades’ oscillation and the discretization of the measurement leads to a slight variation of the circumferential location of the pressure drop, caused by the pressure side of the blade passage, for each blade passage. This variation coupled with the high pressure gradient introduces a high variation of the magnitude between the various pressure traces, inducing a high RMS. The second and the third region, (II) and (III), correspond to the tip leakage vortex. The presence of two high RMS caused by the vortex is explained by the origin of these regions. These are created by the direction of the flow on the sides of the vortex and by the oscillation of the vortex. On the sides of the vortex, the flow is moving nearly perpendicularly to the casing wall, creating two very localized stagnation points on the wall, corresponding to two high pressure gradients. The oscillation of the vortex shifts the position of these high gradient regions, leading to the presence of two high RMS regions. This phenomenon was pointed out by Inoue and Kuroumaru [58].

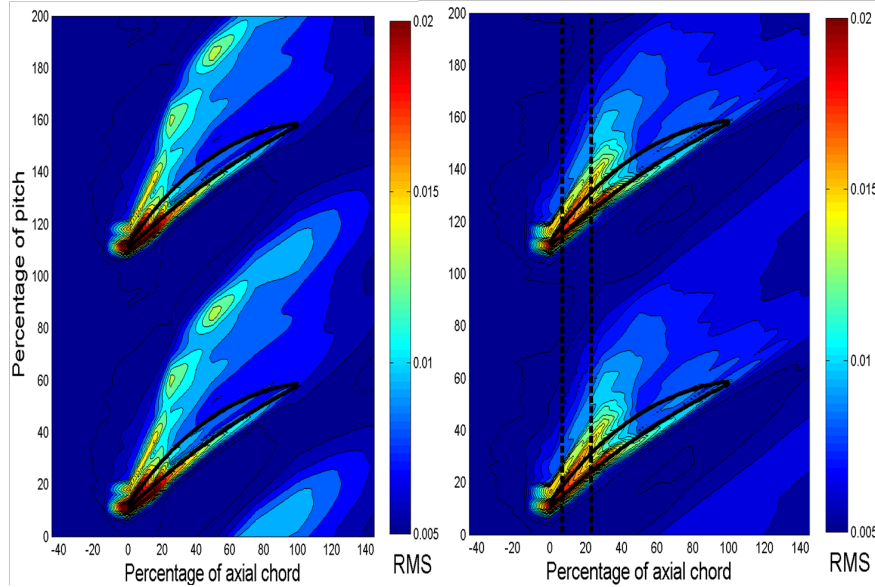


Figure 5.29.: Experimental non-dimensional casing wall static pressure fluctuations (RMS) under near-stall working conditions for the smooth casing configuration (left) and the casing treatment configuration (right)

For working conditions from choke to design, the observation of the RMS field leads to the similar conclusion as from the pressure maps [12]. The more upstream location of the high unsteadiness region corresponding to the vortex is clearly seen to move forward up to the leading edge, and the slope of the vortex axis is steeper as the throttling is increased. The comparison between the RMS field for the

smooth casing and casing treatment configurations shows that the effect of the casing treatment on the tip leakage vortex and the flow field at the tip region is negligible. On the other hand, the RMS visualization allows a better view of the tip leakage phenomenon under near-stall working conditions. The RMS of the wall static pressure at a near-stall working point is presented on Fig.5.29 for the smooth casing and the casing treatment configurations. First of all, the RMS field shows very clearly the trajectory of the tip leakage vortex, better than on the pressure field. Furthermore, the effect of the casing treatment is highlighted by the RMS comparison for both casing configurations. In the case of a smooth casing, the interaction zone of the vortex with the core flow, represented by the high unsteadiness area, covers 75% of the blade passage at 50% of the axial chord. In the casing treatment configuration, the unsteadiness produced by the tip leakage vortex is strongly reduced. This can be explained by the reduction of the rotational intensity of the vortex caused by the fact that the velocity component of the tip leakage flow exiting the groove are negligible perpendicularly to the blade, as presented by Wilke et Kau [114], which leads to the disappearance of a driving force for the rolling-up mechanism of the vortex.

5.3. Simultaneous Acquisitions

In order to investigate the blockage introduced by the tip leakage vortex in the blade passage, as well as the influence of the casing treatment on this blockage, radial traverses with a fast pressure probe were performed downstream the rotor row. The probe used is described in section 5.1.2. The radial traverse counts 16 measurement points between 16.3% to 97.8% of blade height. The phase locked averaging process described previously is applied to the pressure signals from the various radial positions. The pressure traces obtained by the PLA are then used to interpolate the complete map of the total pressure downstream the rotor. The interpolation is performed following the radial and circumferential directions.

The experimental (top) and numerical (bottom) unsteady total pressure field, established from the fast pressure probe, and the casing wall static pressure are presented together on Fig.5.30 for the smooth casing configuration under design working conditions. The total pressure downstream the rotor and the casing static pressure are acquired simultaneously, making this coupled representation of the results convenient. On this figure, the total and static pressures are non-dimensionalized by the inlet total pressure at mid-span, *i.e.* in terms of P_s/P_{01} and P_{03}/P_{01} . In order to capture the various phenomena present in the flow, only the fluctuations of the pressure are presented. This means that the mean value of the pressure at each axial (in casing plane) and radial (downstream the rotor) measuring position has been subtracted from the data measured at this position. Visualization of the three-dimensional flow topology is impossible otherwise, considering the highly non uniform radial distribution of the rotor work. The absolute levels of the casing static pressure and the total pressure downstream the rotor

are too different to be plotted simultaneously with a single scale.

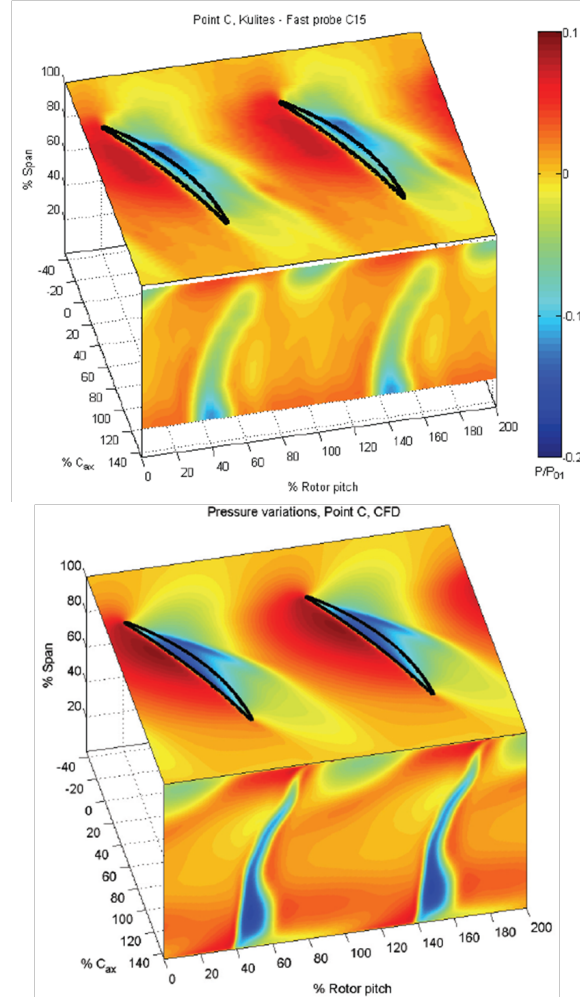


Figure 5.30.: Experimental (top) and numerical (bottom, FINE by B. Farkas) non-dimensional pressure field in the planes of the casing and downstream the rotor, under design working conditions for a smooth casing configuration

The pressure arrangement, used on figure Fig.5.30, gives a very good visualization of the tip leakage vortex path in the two planes and allows highlighting the blockage area in the flow passage introduced by the tip leakage vortex. Furthermore, the blade wake and the hub secondary flow losses and separation are clearly

visible in the plane downstream the rotor. In the casing plane, the low pressure region, corresponding to the origin of the vortex, and the pressure trough indicating the vortex path can be identified, as described previously. The tip leakage vortex introduces a loss core in the plane downstream the rotor. This core is indicated by the nearly circular low pressure region located along the tip region of the flow passage and in the prolongation of the pressure trough in the casing plane. A second area of high losses is seen as a vertical line from hub to around 90% of the blade height. This low pressure area follows the blade profile and corresponds to the wake of the blade. In the hub region, the wake of the blade seems to enlarge. This is due to the superposition of the loss due to the wake and to the separation of the flow at the hub of the blade. Figure 5.31 presents a flow visualization from the numerical results where the tip leakage vortex and the separation at hub are highlighted by using streamlines. The related low pressure regions in the plane downstream the rotor are clearly visible.

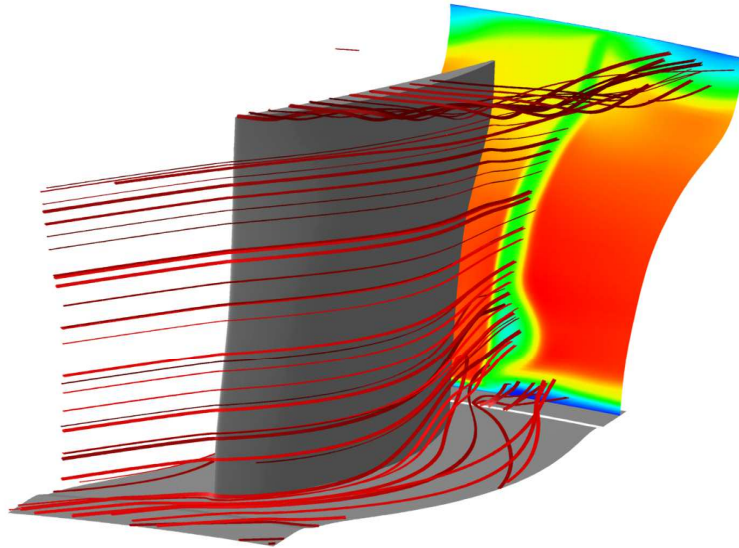


Figure 5.31.: Visualization by numerical streamlines of the tip leakage vortex and the flow separation on the suction side of the blade at the hub under design conditions for the smooth casing configuration, FINE by B. Farkas

The numerical pressure fields are presented on the bottom part of Fig.5.30. The low pressure region corresponding to the phenomena described above are present as well. The tip leakage vortex path can be easily followed in the casing plane. In the plane downstream the rotor, the blockage due to the vortex, the wake of the blade

and the hub flow separation are evidenced. Nevertheless, the comparison with the experimental results reveals some differences. Firstly, the measured thickness of the wake is larger than predicted. This is caused by the non-negligible size of the probe and the related spatial averaging. This spatial resolution leads also to make the separation between the low pressures regions, corresponding to the wake and the tip leakage vortex core, to disappear. In the CFD results, these two regions appear as distinct.

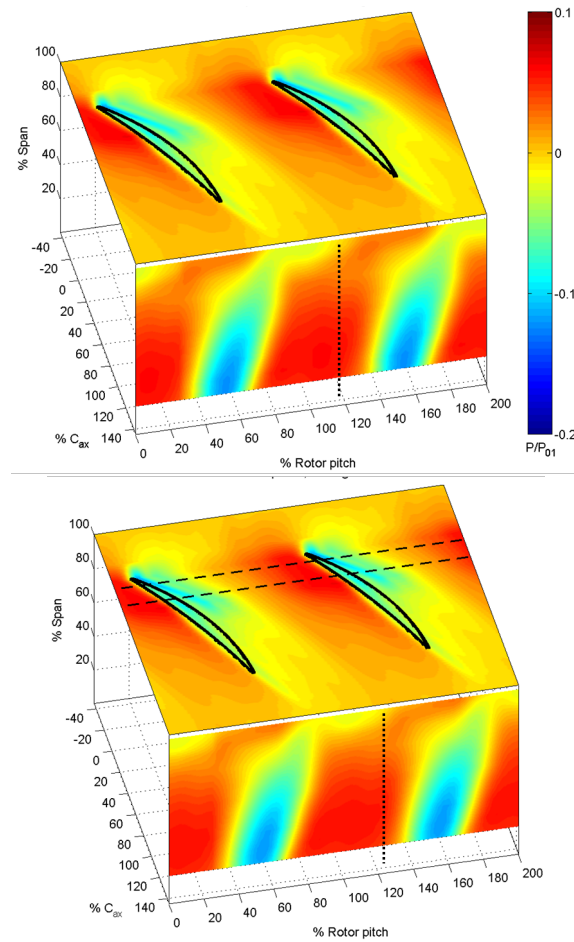


Figure 5.32.: Simultaneous visualization of the non-dimensional measured pressure field in the planes of the casing and downstream the rotor, under near-stall working conditions for a smooth casing (top) and casing treatment (bottom) configurations

This simultaneous visualization of the pressure fields, in the plane of the casing and downstream of the rotor, is used to determine the influence of the casing treatment on the blockage introduced by the tip leakage vortex. As previously observed with the casing static pressure field alone, the circumferential groove casing treatment does not have any influence on the topology of the tip flow and vortex for working conditions from choke (point A) to design (point C). Under near-stall conditions (point D), the effect of the casing treatment are non-negligible. Figure 5.32 presents the experimental pressure field in the planes of the casing and downstream the rotor under near-stall working conditions for a smooth casing (top) and casing treatment (bottom) configurations. The first observation from the study of the pressure field on the plane downstream the rotor is the presence of a very large separation region at the hub of the suction side of the blade indicated by the low pressure. This very large separation at the hub is considered as the cause of the onset of the rotating stall in the compressor. This explains the absence of effect of the casing treatment on the stability margin since this rotor is non tip-critical, as described by Greitzer et al. [43]. Furthermore, the comparison between the pressure fields under design and near-stall conditions shows that the loss core is seen to be weaker as the flow coefficient is reduced.

The separation area at the hub could be caused by a hub corner stall or by a separation of the flow on the suction side of the blade. To determine the origin of the separation, the type of hub corner separation is determined by means of the parameters defined by Yu and Liu [118]. This separation covers 20% of the pitch under design conditions and 35% near stall. Using the characteristics of the compressor and the definition of Yu and Liu, the stall indicator S_w is equal to 0.144 at design working point and 0.252 under near-stall conditions. The diffusion parameter D_m is then calculated from the relative flow angle and velocities presented in section 4.4. The diffusion parameter is 0.203 at design and 0.279 near stall. These values of the stall indicator and of the diffusion parameter show that the compressor presents a hub corner separation under both design and near-stall working conditions. The large separation area at the hub of the blade is thus caused by a flow separation on the suction side of the blade. Figure 5.33 presents a visualization by numerical streamlines of the tip leakage vortex and flow separation at the hub under near-stall conditions. The enlargement of the low pressure region at the hub can be clearly related to the appearance of a blade separation at hub with separated flow, detached from the suction side of the blade. This confirms the corner stall prediction model of Yu and Liu.

The effect of the casing treatment on the tip leakage vortex trajectory is marked by the dashed black lines on Fig.5.32. The presence of the circumferential groove casing treatment induces a circumferential displacement of the vortex core in the plane downstream the rotor. The low total pressure region corresponding to the vortex spreads more into the flow passage in pitch direction for the casing treatment configuration. The radial extension of the vortex does not seem to be affected by the casing groove. A similar influence of the groove is observed in the numerical pressure fields presented on Fig.5.34. On that figure, the overestimation of the

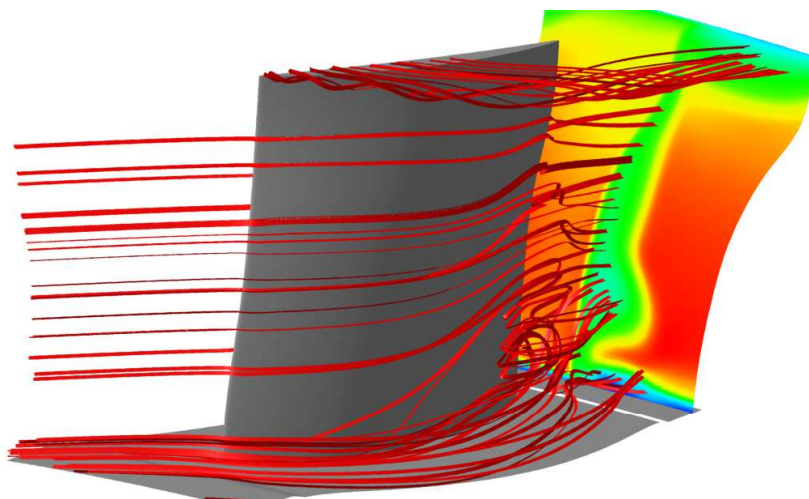


Figure 5.33.: Visualization by numerical streamlines of the tip leakage vortex and the flow separation on the suction side of the blade at the hub under near-stall conditions for the smooth casing configuration, FINE by B. Farkas

extension of the tip leakage vortex core by the CFD can be observed.

In order to get a more explicit view of the extension of the tip vortex in the core flow, we have studied the RMS fields of the casing static pressure and the total pressure downstream the rotor. These RMS fields under choke working conditions for the smooth casing configuration are presented on Fig.5.35. On this figure, the scale of the RMS in the two measurement planes has been adapted in order to improve the visibility. Indeed, due to the structure of the flow, the ratio of the maximum RMS downstream the rotor and the maximum RMS in the plane of the casing is close to 2. The scale of the *RMS* in the casing plane is shown on the figure. In the plane downstream the rotor there are two zones of high unsteadiness. The zone located in the tip region (*i*) corresponds to the core of the tip leakage vortex. The very high level of the unsteadiness in this zone reveals the strong interaction between the vortex flow and the main flow, and thus a high level of loss. The second zone of unsteadiness (*ii*) is caused by the wake of the blade. The relatively low level of unsteadiness in the wake, compared to the vortex core, shows that the principal origin of losses, under these working conditions, is the

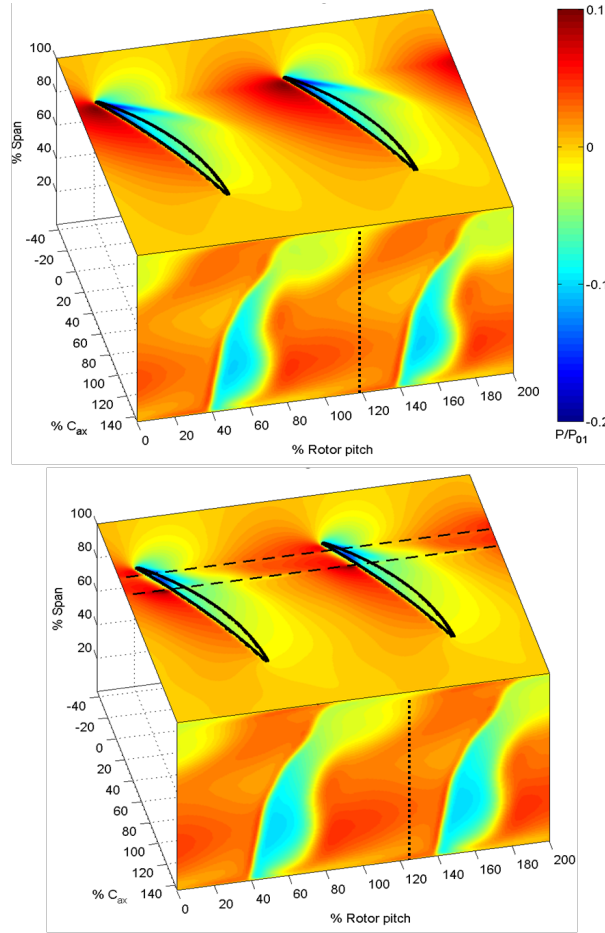


Figure 5.34.: Simultaneous visualization of the non-dimensional numerical pressure field in the planes of the casing and downstream the rotor, under near-stall working conditions for a smooth casing (top) and casing treatment (bottom) configurations, FINE by B. Farkas

tip leakage vortex. The effect of the casing treatment on the tip leakage vortex appears negligible in the RMS field as well. A similar conclusion can be reached under design working conditions.

The RMS fields of the casing static pressure and the total pressure downstream the rotor under near-stall working conditions are presented on Fig.5.36 for both casing configurations. Under these conditions, the relative importance of the two zones in the plane downstream the rotor, the tip vortex core and the wake of the

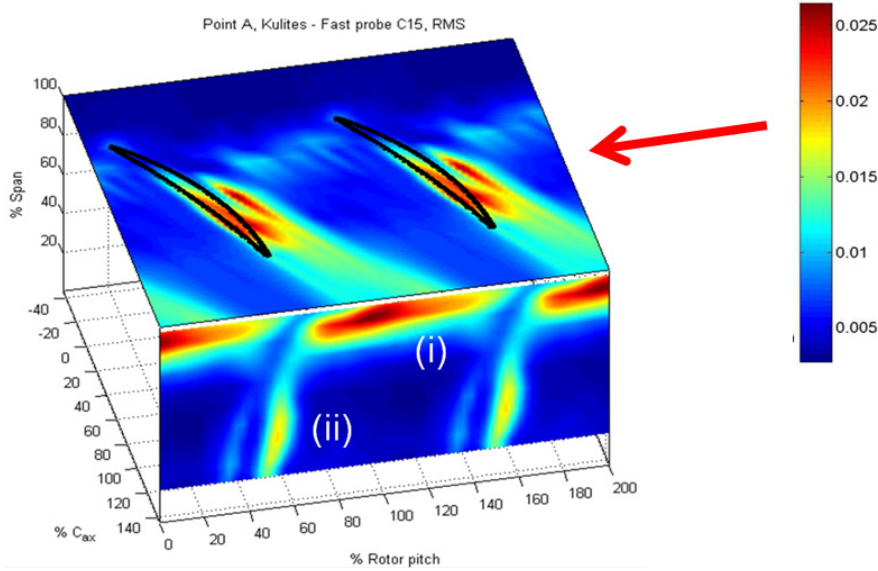


Figure 5.35.: Experimental non-dimensional pressure fluctuations field (RMS) in the planes of the casing and downstream the rotor under choke working conditions for a smooth casing configuration

blade is reversed in terms of unsteadiness magnitude. The unsteadiness level in the wake is higher than in the vortex core, meaning that the principal source of loss is the wake of the blade. The wake unsteadiness zone is actually caused by the wake of the blade itself mixed with the hub blade separation. The inversion of the relative importance of the two zones is caused by an increase of the unsteadiness in the wake, due mainly to the apparition of a hub corner stall, and by a reduction of the unsteadiness level in the tip leakage vortex core. This confirms the decrease of the weakness of the loss core, observed in the pressure fields, as the flow coefficient is decreased.

Concerning the effect of the casing treatment on the pressure fluctuations field, the circumferential displacement of the vortex core, due to the presence of the groove, is equally observed in Fig.5.36. The red line indicates the extension of the tip leakage vortex core in the flow passage at 98% of the blade height. With the smooth casing, the vortex core extends from 100% to 130% of the pitch. In the casing treatment configuration, the extension of the vortex is between 100% to 140% of the pitch. Furthermore, the RMS in the tip vortex core shows a lower value with casing treatment. This signals a lower level of unsteadiness in this region, inducing a reduction of the interaction between the vortex and the core flow. The displacement of the vortex core and the reduction of the unsteadiness are shown in

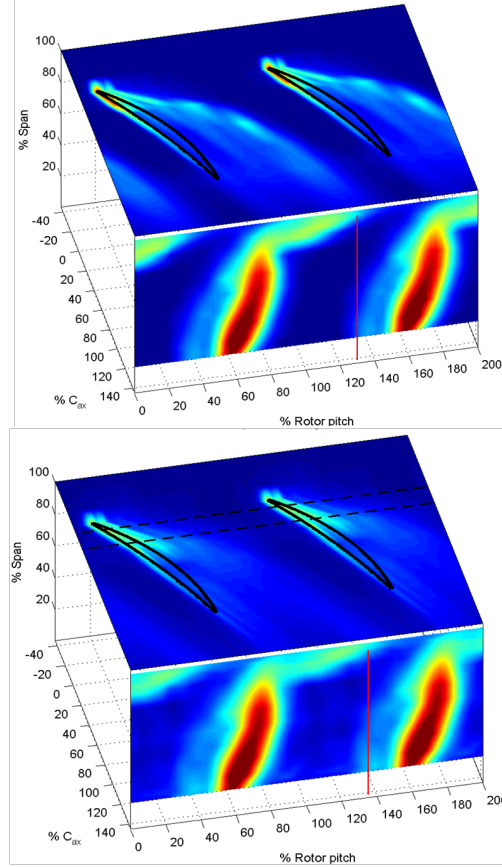


Figure 5.36.: Experimental non-dimensional pressure fluctuations field (RMS) in the casing and downstream rotor planes under near-stall working conditions for a smooth casing (top) and casing treatment (bottom) configurations

Fig.5.37. This figure shows the difference of the RMS fields between the two casing configurations $RMS_{smooth} - RMS_{treatment}$. The high positive difference in the tip region (i) indicates a lower unsteadiness in the casing treatment configuration. This shows the reduction effect of the casing treatment on the unsteadiness. The high negative region, located just to the right (ii), is caused by the increase of the circumferential extension of the vortex core in the case of the casing treatment.

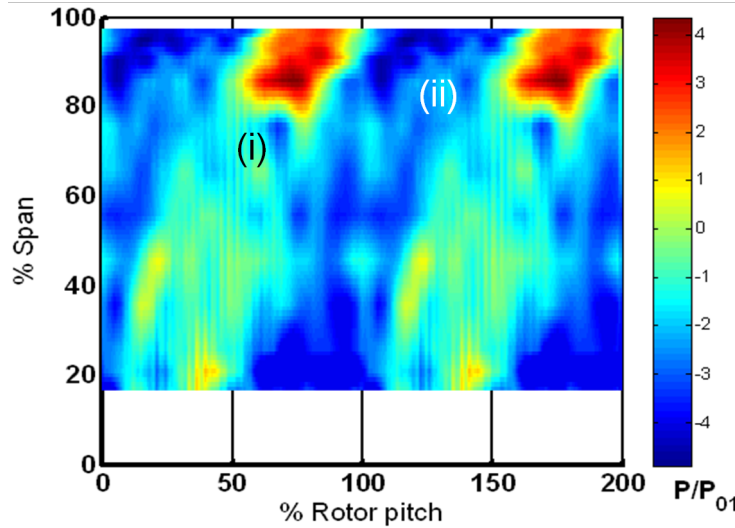


Figure 5.37.: Difference of the experimental non-dimensional total pressure fluctuations (RMS) downstream the rotor between the two casing configurations under near stall working conditions

5.4. Conclusion

This chapter presents the results of the investigation of the three-dimensional unsteady phenomena occurring in the rotor row of the VKI R4 low-pressure axial compressor. Furthermore, the effect of a circumferential groove casing treatment on the unsteady flow field in the tip region of the compressor is shown. The flow fields are presented in terms of casing wall static pressure and total pressure downstream the rotor. The experimental pressure measurements are done by means of a specific instrumentation. This instrumentation is characterized by a very fast response and a high level of miniaturization. The casing wall static pressure is measured by using fast sensors, placed in the casing. A system of inserts is used in order to protect the sensor and to increase the spatial resolution of the measurements. The inserts are flush mounted in the casing at various axial locations in order to cover from -45% to 145% of the axial chord and they are spaced by one rotor pitch in the circumferential direction. Their axial final position allows avoiding the influence of the sensor wake on the measurements. The frequency analysis shows that the use of the inserts reduces the frequency domain of the measurement system. Nevertheless, this frequency domain being around 12 times the blade-passing frequency, it remains largely sufficient.

A post-processing procedure has been developed and applied to the pressure signals in order to extract the information at the time scale of a blade passage.

This procedure allows to build the pressure field over a single pitch and over the complete measurement axial domain from the pressure signals from nine sensors, acquired during approximately 16 revolutions. The first part of the post-processing is the reduction of the pressure signal corresponding to 1377 blades passages to a mean signal over a single blade-passage. This method is called the Phase Locked Average, or PLA. The first step of the PLA is the decomposition of the entire signal in signals corresponding to each blade passage. This is based on a one-per-revolution trigger signal and assumes that the rotational speed of the rotor is constant over one revolution. Then the pressure signals of each blade passage is interpolated in order to obtain signals with a constant number of points and points positions. Finally the various pressure signals are averaged point by point and the resulting signal is the mean signal over one blade-passage. The PLA of the pressure signals from the various sensors are used to interpolate the pressure field over one pitch and over the complete measurement axial range. Due to the very low resolution of the measurement in the axial direction compared to the resolution in circumferential direction, specific directions of interpolation have to be considered.

The interpolated unsteady casing wall pressure fields show clearly the important phenomena in the flow in the tip region. In these fields, a low pressure region, corresponding to the origin of the tip leakage vortex, is visible along the suction side of the blade. The path of the tip vortex in the blade passage is marked by the low pressure zone extending from the low pressure region along the blade and detaching from the blade. These two elements allow a good visualization of the position of the tip leakage vortex as well as its direction. Finally, the pressure side of the blade corresponds to a very sharp pressure gradient. The origin of the tip vortex is seen to move towards the leading edge of the blade as the flow coefficient is reduced. Furthermore, the reduction of the mass flow leads to an increase of the slope of the axis of the tip vortex. These two observations confirm previous investigations performed in this field.

Under near-stall working conditions, the origin of the tip leakage vortex is located at the leading edge of the blade. The trajectory of the tip vortex is seen to stay into the blade passage while remaining quite far from being tangentially aligned with the plane of the leading edges of the rotor. Furthermore, we can notice the absence of any backflow from tip clearance flow at the trailing edge of the blade and of any spillage of the tip clearance flow below the blade tip of the adjacent blade. This is related to the typical modal stall inception process. This stall inception mode of this compressor was already pointed out in section 4.3 by the nearly zero-slope of the total-to-static characteristic at the stability limit.

The effect of the Reynolds number on the wall static pressure field has also been investigated. This effect is negligible except for a circumferential shift of the blade position relative to the trigger position due to an increase of the torque at high Reynolds numbers.

The comparison between the experimental results and the numerical simulations shows a high similarity. The three noticeable regions identified above are clearly

visible in the CFD pressure field as well. The position of the low pressure region, indicating the origin of the tip leakage vortex, is well predicted by the FINE/turbo code under the various working conditions. Nevertheless the direction of the axis of the tip leakage vortex in the blade passage presents a steeper slope in the CFD results, and this under all working conditions. This can be explained by a difference in terms of tip gap between the CFD simulations and the real tip gap in the test section.

The influence of a circumferential groove casing treatment on the wall static pressure field has been investigated. The results show that the casing treatment has a negligible effect on the flow outside the groove area when the origin of the tip leakage vortex is located downstream the casing treatment. Under near-stall working conditions, corresponding to the case where the origin of the tip vortex is located upstream the treatment, the circumferential groove leads to a reduction of the slope of the vortex axis with respect to the axial direction, and to a reduction of the extension of the vortex in the casing wall plane due to a radial displacement of the vortex towards the hub. These observations are confirmed by the numerical simulations.

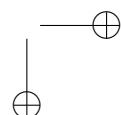
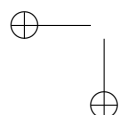
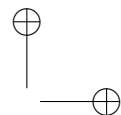
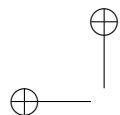
The pressure fluctuation fields, represented by the RMS, give a very good visualization of the tip leakage vortex. In the RMS, the tip vortex is characterized by a high unsteadiness level due to the high level of turbulence, and to the high pressure gradient coupled with the oscillation of the phenomenon. The tip vortex thus appears as two regions of high unsteadiness corresponding to the sides of the vortex. On the RMS field, a third region of high unsteadiness, corresponding to the pressure side of the blade, is visible. The comparison of the RMS fields for the two casing configurations confirms the effect of the circumferential groove on the tip leakage vortex path.

The simultaneous presentation of the casing wall static pressure and the total pressure downstream the rotor allows a visualization of the tip leakage vortex path in the three dimensions, as well as a visualization of the hub blade separation. In the measurement plane downstream the rotor, the total pressure field presents three low-pressure areas. The first appears as a nearly circular area located in the tip region. This corresponds to the tip leakage vortex core. The second area extends from the hub to close to the tip and is due to the wake of the blade. The last area, located in the hub region along the suction side of the wake, is caused by the flow separation on the suction side of the blade.

In the plane downstream the rotor, the circumferential groove casing treatment leads to a modification of the position and of the intensity of the tip leakage vortex core. This modification is only visible under the near-stall conditions as well. The casing treatment introduces a circumferential displacement of the tip vortex core towards the adjacent blade of 10% of the pitch at 98% of the blade height. Furthermore the intensity inside the tip leakage vortex core is reduced by the presence of the casing treatment. This is confirmed by the CFD results and by the RMS fields.

Under near-stall working conditions, the low pressure area in the plane down-

stream the rotor, corresponding to the hub blade separation, grows sharply until covering 35% of the pitch. The determination of the hub corner stall parameters introduced by Yu and Liu [118] shows that the rotor presents a hub corner separation at both design and near stall. This very high source of blockage and loss is considered to be the origin of the instability in the compressor.



Chapter 6.

Stall Characterization

This chapter focuses on the rotating stall phenomenon occurring in the compressor studied in the present thesis. This study is based on the measurement of the unsteady wall static pressure at the rotor casing and on the measurement of the main flow velocity. The specific instrumentation composed of fast pressure sensors and hot-wires is presented in the first section. The description of the rotating stall inception process given above evidences that the compressor presents a modal stall inception with full-span stall cells extending over the entire stage. The evolution over time of the rotating stall with a cell-splitting phenomenon is described. Furthermore a complete characterization of the rotating stall cells is performed in terms of number, rotational speed and size of the stall cells.

6.1. Specific Instrumentation

The study of the stall inception and the characteristics of the stall cells is based on the measurement of the unsteady wall static pressure at the rotor casing, using fast pressure sensors, and the determination of the velocity variation in the main flow at different axial and circumferential positions by the use of hot-wires. The fast pressure sensors are those used for the tip leakage vortex investigation and presented in Section 5.1.1. For this investigation, we have used three of them, placed at the same axial position but at three different circumferential positions. These sensors, called $K7$, $K10$ and $K11$, are located at 15% of the chord upstream of the leading edge of the rotor blade. In the circumferential direction, the sensors are located at 0, corresponding to 120 deg and 220 deg as presented on Fig.6.1. They are thus not equally spaced around the annulus. This particular positioning gives the opportunity to investigate the number and velocity of stall cells by measuring the phase of pressure oscillation.

In addition to the fast pressure sensors, four hot-wires are used for the stall characterization. All of them are placed at the same circumferential position. In the axial direction, they are placed at the positions of the four measurement planes corresponding to upstream stage, upstream rotor, downstream rotor and downstream stage. The hot-wires are fixed on the radial displacement carriages presented in Section 5.1.1. This arrangement, shown on Fig.6.2, allows determining the radial and axial extension of the stall cells in the test section.

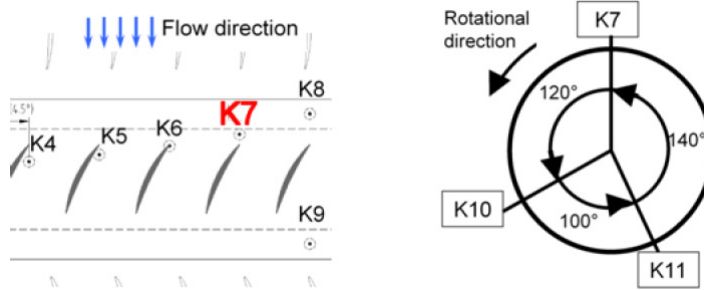


Figure 6.1.: Axial and circumferential positioning of the fast pressure sensors used for stall cell detection

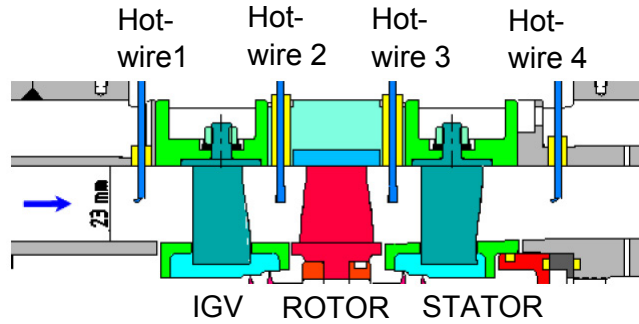


Figure 6.2.: Axial arrangement of hot-wires used for stall cell detection and characterization

6.2. Stall Inception Process

In order to study the rotating stall inception process, the compressor is brought to the near-stall operating point (point D) presented in Section 3.5. This point is the closest to the stability limit. Then the compressor is progressively pushed to the unstable regime by closing the throttling valve. When the stall occurs, the valve position is maintained constant. The signal acquisition from fast pressure sensors and hot-wires takes place during this operation. Figure 6.3 shows the raw voltage signal from fast pressure sensor *K7* acquired during a stall inception process in a smooth casing configuration. On this figure, the time axis is given in terms of rotor revolutions. Two different working regimes can be observed. In the first part of the signal, before revolution 300, the compressor is running at stable conditions. The variation of the voltage, visible on the figure, is due to the pressure change

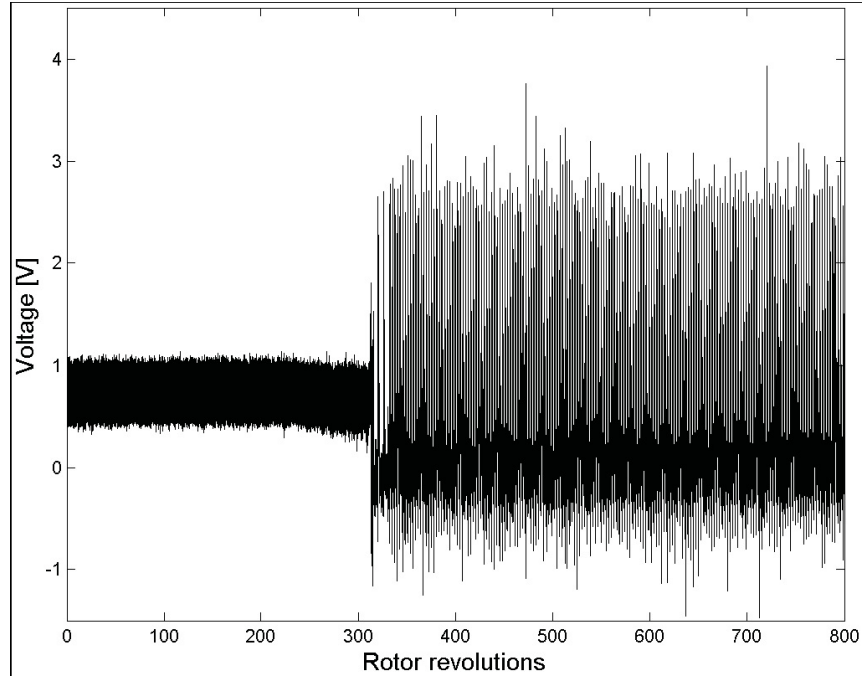


Figure 6.3.: Fast pressure sensor K7 voltage raw signal during stall inception process for smooth casing configuration.

over one pitch. Around revolution 300, oscillations of high amplitude appear in the pressure signal. At this point the compressor comes out of stable operating conditions and comes in rotating stall. The high amplitude oscillations correspond to the stall cells.

A zoom on the signal from fast pressure sensors *K7*, *K10* and *K11* around revolution 300 is presented on Fig.6.4. This zoom allows a clear visualization of the stall inception process. The traces from the sensors are plotted one below the other with respect to their circumferential position, and the signal from sensor *K7* is duplicated at the bottom. This arrangement allows observing the annular evolution of the stall inception process in function of the time. On this figure some disturbances in the signal from all the sensors are visible during the twenty last revolutions before the unstable regime is reached.

Applying a low-pass filter at 30 *Hz* on the signal allows showing this phenomenon. Figure 6.5 presents the filtered signal from the three fast pressure sensors for smooth casing configuration. The disturbances are clearly revealed by the filtering and appear like modulations superimposed on the pressure signal. Around revolution 310 the amplitude of these modulations increases abruptly

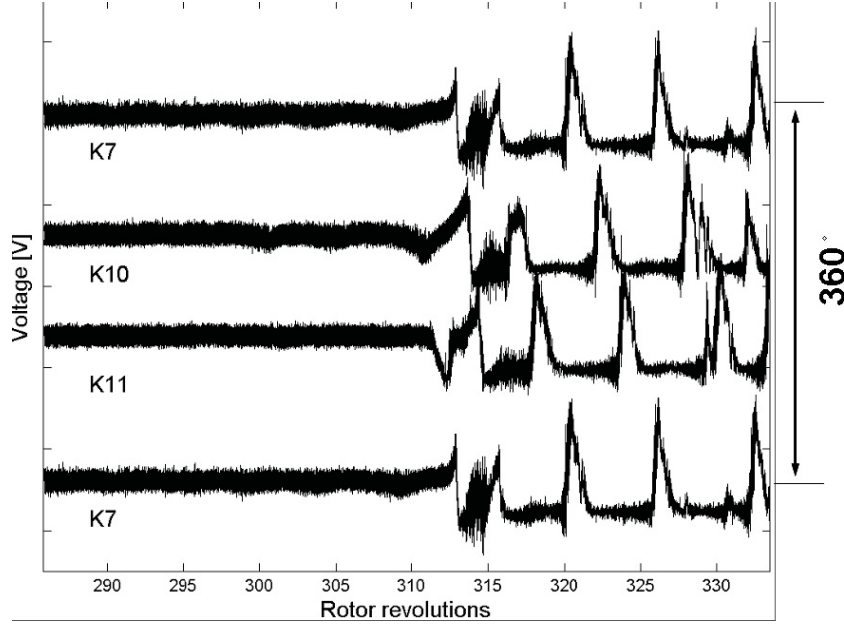


Figure 6.4.: Voltage raw signal from fast pressure sensor located at different annular position during stall inception process for smooth casing configuration.

to turn into unstable regime with low voltage regions separated by high voltage peaks. The modulations observed in the signals are presented by Camp et al. [15] as the sign of the presence of a "modal wave stall inception". This is coherent with the behavior of the characteristics of the flow in the compressor under near-stall conditions. Firstly, the unsteady casing pressure measurements, presented in Chapter 5, showed the evolution of the tip leakage vortex trajectory as the flow coefficient is reduced. Under near-stall conditions, this trajectory is still going well into the blade passage and remains quite far from being in the leading edge plane. No signs were observed of any backflow from the tip clearance fluid at the trailing edge or any leading edge spillage of tip clearance flow below the blade tip to the adjacent blade. Secondly, in chapter 4, the compressor is shown to reach the stability limit at the zero-slope point of the total-to-static characteristic. According to Tan [106], these behaviors under near-stall conditions are characteristic for compressors presenting modal wave stall inception.

The unstable regime shown in Fig.6.5 is typical of the compressor rotating stall phenomenon. The low voltage region corresponds to a high static pressure and then to a low speed area related to a stall cell. The stall cells are separated by high speed regions corresponding to low static pressure represented by the high

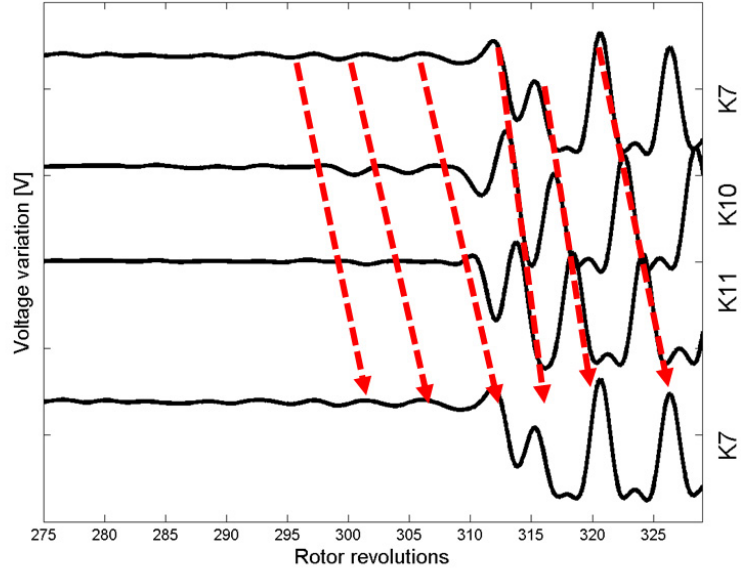


Figure 6.5.: Low-pass filtered voltage signals from fast pressure sensors located at different annular positions during stall inception process for the smooth casing configuration.

voltage peaks. The plot shows clearly the rotating character of the instability phenomena. The signals show a phase shift of the modulations and the stall cells between the various fast sensors. This shift is revealed by the red dotted lines on the figure. That implies a rotation of these phenomena around the annulus in the same rotational direction as the rotor. The unstable phenomena occurring in this compressor are rotating stall and not surge.

Similar measurements were performed for the casing treatment configuration. The pressure signals from the three fast sensors, low-pass-filtered at 30 Hz , are presented in Fig.6.6. An identical behavior in terms of modal wave stall inception and rotating stall cells can be observed.

The radial and axial extension of the stall cells can be extracted from the study of the hot-wires put at different positions in the test section, as presented in the previous section. This arrangement allows a visualization of the stall inception process over the entire stage. Figure 6.7 presents the voltage signals around the stall inception point from the four hot-wires situated at hub for upstream stage and rotor, and at tip for downstream rotor and stage. The red line indicates the first pressure variation due to the stall and the green line the first completely

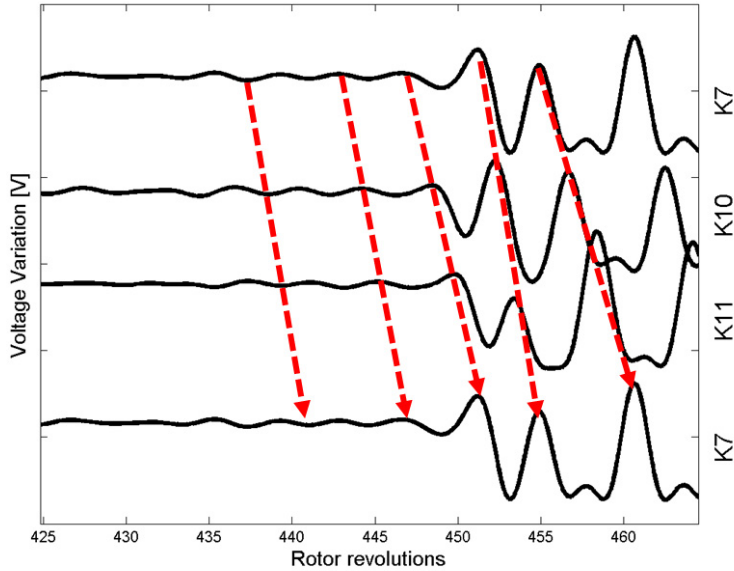


Figure 6.6.: Low-pass filtered voltage signal from fast pressure sensors installed at different annular position during stall inception process for the casing treatment configuration.

formed stall cell. These two events occur on the four traces exactly at the same time. This implies that, from its inception, the stall cells are full-span stall cells with an axial extension over the entire stage.

Furthermore, the stall presents a similar pattern over the entire height of the blade. This is not always the case. For example, Durmusoglu [33] investigated the stall cell pattern in a low pressure compressor and observed a different behavior at mid-span and at tip. The author suggests that the stall was more severe at tip. But this investigation was performed on a compressor with a blade diameter three times higher than in the work presented in this thesis.

6.3. Rotating Stall Patterns

6.3.1. Stall Cells Time Evolution

During several experiments concerning the rotating stall pattern it was observed that the behavior of the compressor in unstable regime evolves over time in each experiment. This pattern was also changing between the various experiments

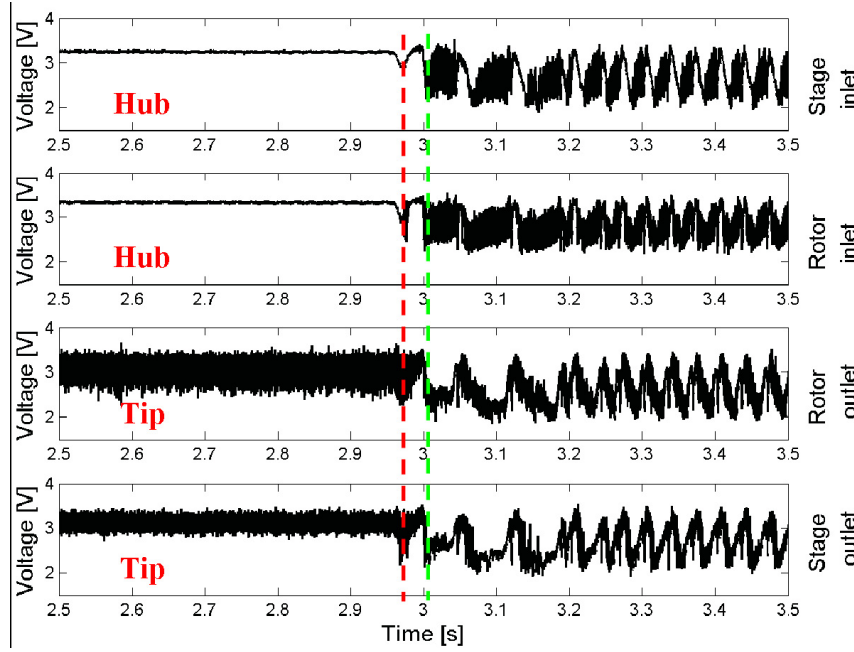


Figure 6.7.: Voltage signal around the stall cell inception point from hot-wires.

performed under identical conditions. This is shown in Fig.6.8 where we present the voltage signals of Kulite K7 for three different experiments. The signal from each experiment can be divided in three parts belonging to three different operating regimes. The first part (*I*) corresponds to the stable regime of the compressor and stretches over the first second of the signal. In the second (*II*) and in the third (*III*) part, the compressor operates in unstable regime. The difference between these two parts comes from the variation of the frequency of the rotating stall phenomenon. This frequency is higher in the third part than in the second.

The difference between the second and third parts shows the evolution over time of the compressor in unstable regime. In order to determine the nature of this evolution, Fig.6.9 show a zoom on the passage from part *II* of the signal to part *III*. In part *II* of the signal, equally-spaced stall cells are clearly visible. In the transition region, a small cell appears to the right of the main stall cell, as indicated by the red circle. At the next passage of the stall cells, two cells are visible side-by-side and begin to move apart from one another. After two supplementary cell passages the transition is finished and the signal presents similar and equally-spaced stall cells. The time evolution of the rotating stall is a splitting of the cells.

This phenomenon of variation of the number of stall cells over time has been

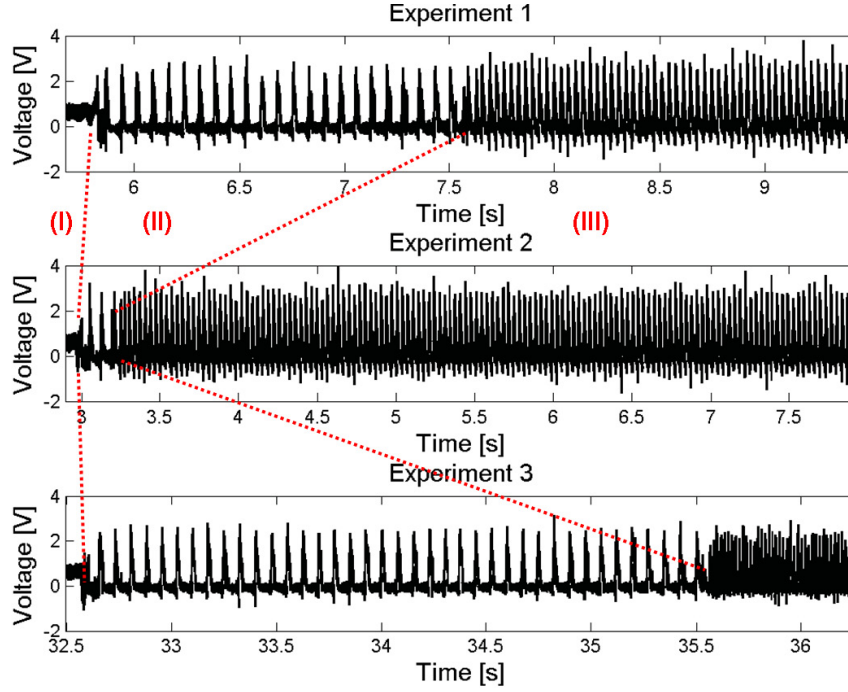


Figure 6.8.: Raw voltage signal from fast pressure sensor *K7* for three different experiments during rotating stall in smooth casing configuration.

observed by several authors for a large range of compressors in terms of rotational speed, hub-to-tip ratio. This variation can be an increase or a decrease of the number of stall cells, and this evolution can be divided in several steps. In 1952, Huppert [57] characterized experimentally the stall cells configurations in a single-stage axial compressor for various hub-tip ratios. For a ratio of 0.5, the compressor presented 3, then 4 and finally 5 part-span stall cells. In the case of a hub-tip ratio equal to 0.8, the compressor moves from a configuration with 8 part-span stall cells to a single full-span stall cell. Day and Cumpsty [28] studied the impact of the rotation speed on the stall configuration. They noticed a variation of the number and the type of stall cells from 4 part-span stall cells to 12 part-span stall cells, then 1 full-span cell. At intermediate speed the authors observed a configuration with 1 part-span stall cell developing in full-span cell. In 1996, Poensgen and Gallus [87] described the process of the passage from a single stall cell to a two-cell configuration in a single-stage axial flow compressor. This process was also numerically observed by He [49] on a single blade-row with an evolution from 5 to 7 cells.

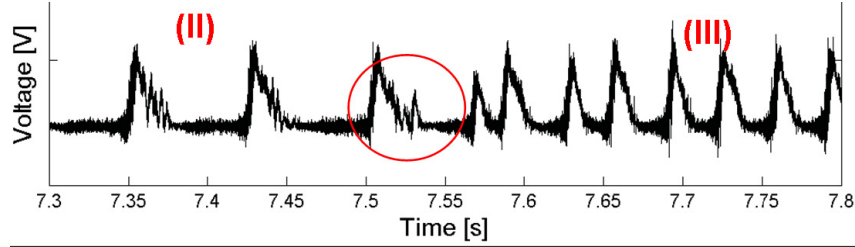


Figure 6.9.: Zoom on the time evolution of the raw voltage signal from fast pressure sensor *K7* in smooth casing configuration.

The variation of the unstable operating regime between the various experiments can be observed on two parameters. The first is the duration of the second part of the signal. In experiment 1, this part stretches over about 1.7 *sec* while it stretches over around 0.2 *sec* in the second experiment and 3 *sec* in the third. The second parameter is the frequency of the rotating stall phenomena in part *III* of the experiments. In experiments 1 and 2, this frequency seems to be equal but higher in experiment 3. The quantitative study of these variations can be performed using a frequency analysis of the signals through the Fourier and wavelet analysis.

In order to determine the frequency of the different phenomena occurring in the unstable operating regime, it is necessary to establish the frequency spectrum of rotating stall. This spectrum is based on a Fourier analysis. This method was often used for the investigation of rotating stall by *i.e.* Levy et al. [71], Levy and Pismenny [70], Gannon et al. [39] and Mailach and Vogeler [79]. The Fourier transformation for a discrete signal x_k of length N , acquired at a frequency F_{acq} is defined as (Tropea [109]):

$$X_n = \sum_{k=0}^{N-1} x_k e^{(-\frac{2\pi i k n}{N})} \quad (6.1)$$

where X_n are the complex Fourier coefficients resulting from the Fourier transformation. The power spectral density (PSD) is commonly used to present the frequency content of a signal. This density is defined for a discrete signal as:

$$PSD(f_n) = \frac{\Delta t}{N} |N_n|^2 \quad (6.2)$$

where the time step of the acquisition Δt and the frequency f_n are given by:

$$f_n = \frac{n}{N\Delta t} \text{ with } n = 0, 1, \dots, N-1 \quad (6.3)$$

$$\Delta t = \frac{1}{F_{acq}} \quad (6.4)$$

A Fast Fourier Algorithm (FFT) is used to calculate the Fourier coefficients. Thus N must be chosen as $N = 2^p$. The frequency resolution of the FFT is calculated with the following equation:

$$\Delta f = \frac{1}{N\Delta t} = \frac{F_{acq}}{N} \quad (6.5)$$

The highest frequency f_{max} that can be resolved is equal to:

$$f_{max} = \frac{1}{2\Delta t} = \frac{F_{acq}}{2} \quad (6.6)$$

The Fourier analysis makes it possible to present a signal in the frequency domain but with a total loss of the time information. The example of experiments presented on Fig.6.8 shows that the time is an important parameter in the evolution of the rotating stall process. The study of stall cell behavior requires to determine the frequency of the occurring phenomena, as well as its evolution over time. The short time Fourier transformation allows a representation of a signal in the frequency domain in function of the time. This method is based on the division of the signal in time segments, called windows. The short time Fourier transformation is performed on each signal segment. This frequency signal analysis regarding the study of rotating stall process was used by Gannon [39]. The problematic part of this method is the choice of the time-window's size. A small window leads to a good resolution in the time domain but a poor resolution in the frequency domain and the other way around.

An alternative way is to use a Wavelet transformation (WT) that provides a good frequency resolution at high frequencies and good time resolution at low frequencies by using a variable window size. The wavelet coefficient of the continuous WT at scale k and time t is defined as (Lewalle [72]):

$$W(k, t) = \sqrt{k} \int_{-\infty}^{\infty} f(y)g(k(y-t))dy \quad (6.7)$$

In this equation $f(y)$ is the signal in the time domain and $g(z)$ is the so called “wavelet” function with $z = k(y - t)$. The wavelet function performs a windowing of the signal around time t with an expansion parameter k . Equation 6.7 is a convolution of the signal $f(y)$ and the wavelet $g(z)$. It means, if the signal is constant near t , that the wavelet coefficient will be zero. If, at t , the signal has the same shape as the wavelet, it will result in a large number of wavelet coefficients. A very large value of function f in the time domain will imply a large value of the wavelet coefficients. By changing time t in equation 6.7, the complete signal is scanned. The wavelet will dilatate or shrink by varying the parameter k .

Numerous shapes and analytical forms of wavelets can be found in the literature. Lewalle [72] identifies the admissible conditions for a function to be a wavelet. Two examples of common wavelet functions used for wavelet transformation are the “Mexican Hat” (Fig.6.10) and the complex “Morlet” wavelet (Fig.6.11).

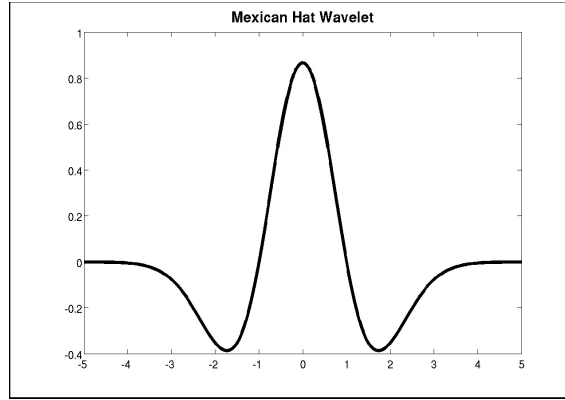


Figure 6.10.: Shape of Mexican hat wavelet.

These functions are analytically defined as respectively:

$$g(z) = (z^2 - 1)exp(-z^2/2) \quad (6.8)$$

$$g(z, z_0) = (\cos 2\pi z + i \sin 2\pi z)exp(-2z^2\pi^2/z_0^2) \quad (6.9)$$

Choosing the right wavelet is an important decision while dealing with the WT. For example the Morlet wavelet provides a better spectral resolution than the Mexican hat, but with a poorer time resolution. In this thesis, a complex Morlet wavelet is used, because it provides sufficient accuracy in time and frequency resolution (Lin et al. [74]). We use a spectral density for the presentation of the results. It is defined as (Tropea [109]):

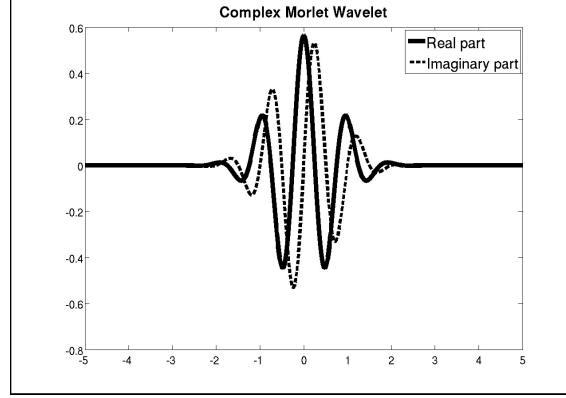


Figure 6.11.: Shape of complex Morlet wavelet.

$$PSD = |W(k, t)|^2 \quad (6.10)$$

As an example the following intermittent signal is considered:

$$\begin{aligned} y &= 5 & \text{for } 0 \text{ s} < t \leq 2 \text{ s} \\ y &= 5 + \sin(6 \pi) & \text{for } 2 \text{ s} < t \leq 3 \text{ s} \\ y &= 5 & \text{for } 3 \text{ s} < t \leq 4 \text{ s} \\ y &= 5 + \sin(12 \pi) & \text{for } 4 \text{ s} < t \leq 5 \text{ s} \\ y &= 5 & \text{for } 5 \text{ s} < t \leq 6 \text{ s} \\ y &= 5 + \sin((2\pi)^2) & \text{for } 6 \text{ s} < t \leq 10 \text{ s} \end{aligned}$$

Figure 6.12 presents this function in the time domain (above) and the wavelet map of the signal (below). In this map, the axis of abscissas corresponds to the time and the axis of ordinates to the frequency. This plot shows that a region of constant value of the signal results in a zero value of the PSD. After two seconds the map shows a bright area extending over the time of the sine signal. The frequency in the wavelet map corresponds to the frequency of the sine wave. The same behavior is reflected in the second sine signal, however with a higher frequency. After six seconds, a bright area starts out at nearly one Hz and increases with time, due to the increasing frequency of the signal. This example shows the possibility to determine the time-dependent frequency spectrum of a signal allowed by the wavelet transformation. An investigation of rotating stall and the stall inception process with a WT is presented in Lin et al. [74], [75] and [76]. Further readings about wavelet analysis can be found in Lewalle [72].

Regarding the three experiments presented below, the pressure signals were acquired with a sampling frequency of 50 kHz over minimum 15 sec, giving a Fourier

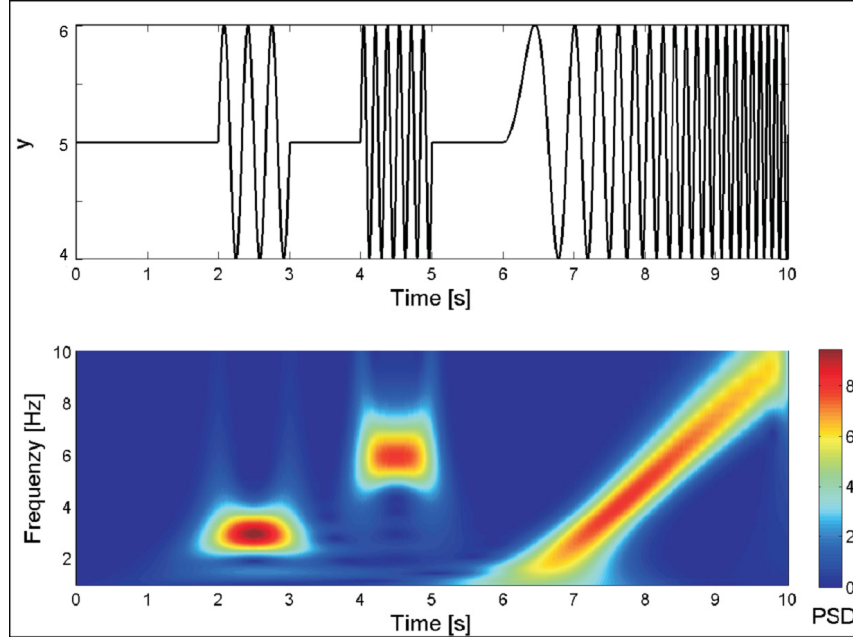


Figure 6.12.: Intermittent signal (above) and its wavelet transformation (below).

transform maximum frequency of 25 kHz and a frequency resolution of minimum 0.1 Hz . Figure 6.13 and Fig.6.14 show the low frequency part of respectively the power spectral density and the wavelet transformation of the pressure signals from fast sensor *K7* for the three experiments.

In the case of experiment 1, two frequency peaks at 13.4 Hz and 29.4 Hz are visible as well as the corresponding harmonics in the PSD. These frequency peaks and their harmonics are also visible on the wavelet transformation of the signal, in red for the principal frequencies, and in light blue for their harmonics. The peak at 13.4 Hz appears to be corresponding to part *II* of the signal, and the peak at 29.4 Hz to part *III* as defined at the beginning of this chapter.

The PSD of experiment 2 presents a peak at 29.3 Hz as well as a very low peak between 12 Hz and 16 Hz . With the wavelet map it can be evidenced that the low frequency at the beginning, corresponding to part *II* of the signal, corresponds to the very low peak and the next frequency to 29.3 Hz . The very low amplitude in the PSD of the frequency peak corresponding to part *II* is due to the very short duration of this part (about 0.3 sec) in comparison with the duration of the signal considered for the PSD determination.

Combined examination of the pressure signal, the FFT content and the wavelet map for Experiment 3 reveal an initial rotating stall frequency of 13.4 Hz . The

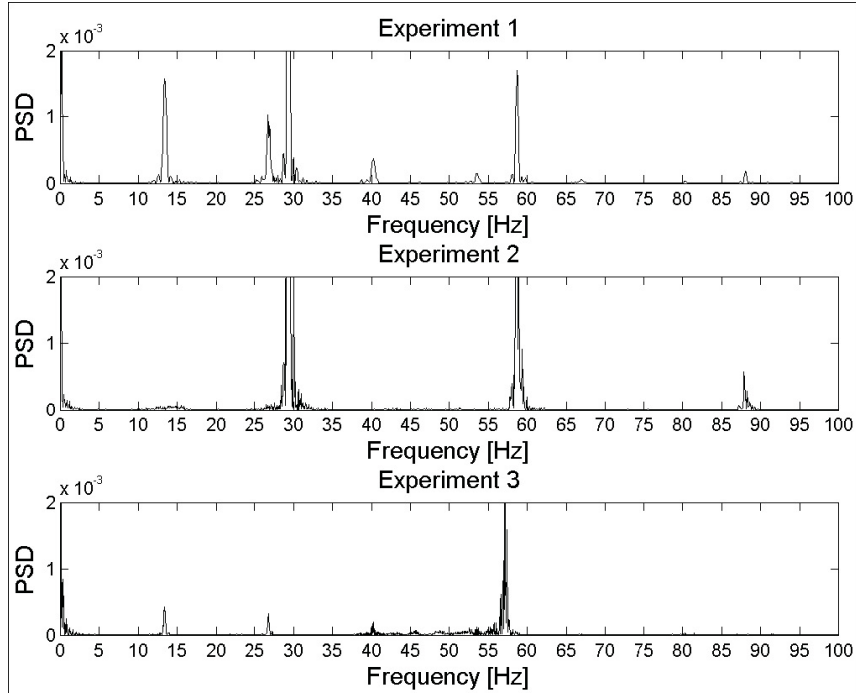


Figure 6.13.: Spectral density of the fast pressure signal for three experiments.

higher frequency of rotating stall occurring at 35.5 sec is 57.1 Hz.

These examples show the importance to combine the two frequency analyses in order to extract a maximum of information about the unsteady phenomena of rotating stall from the time signal. The wavelet transformation allows to establish the time evolution of the frequency of the stall cell as well as to determine all the frequencies present in the signal. The determination of these frequencies with a high accuracy is performed based on the PSD. The three examples presented here correspond to a smooth casing configuration. Experiments have also been done for the circumferential groove casing treatment and the same time evolution of stall cell is observed.

6.3.2. Stall Cells Characteristics

The frequency analysis of the pressure signals presented in the previous section shows a time evolution of the rotating stall regime. This regime can be divided in two time areas characterized by the frequency of the unsteady phenomena. In this section the characteristics of the rotating stall cells of the different parts and

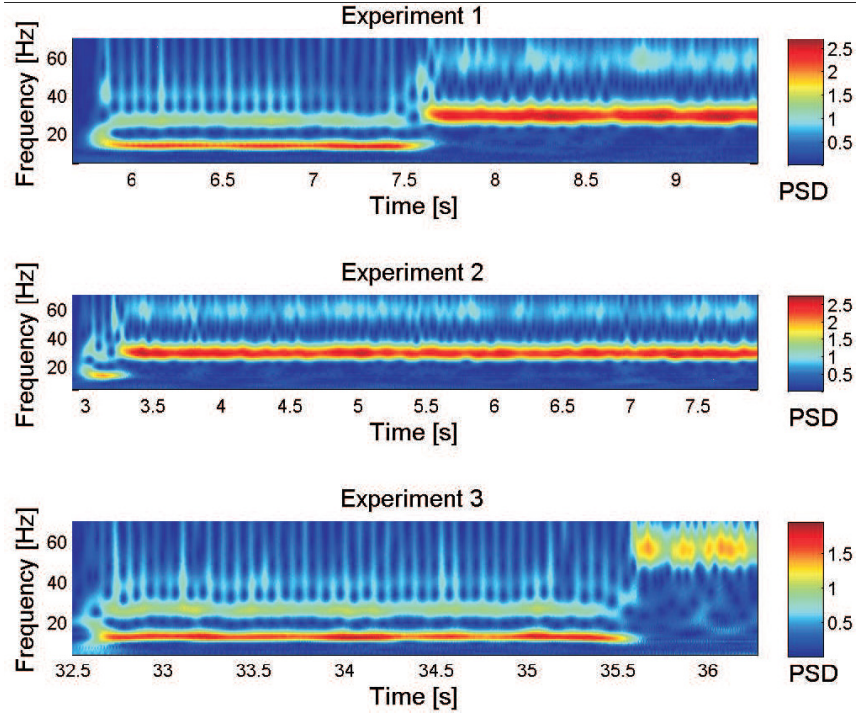


Figure 6.14.: Wavelet transformation of the unsteady pressure signal for three experiments.

experiments are presented in terms of number of cells, size and rotating speed of cells.

The determination of the number of stall cells in the compressor is based on the method suggested in Levy [70]. An idealized signal y of the static pressure in a compressor during stall can be described as a simple sine function:

$$y = A \sin(\omega_{OSC}t + \varphi) \quad (6.11)$$

where A is the amplitude of the signal, ω_{osc} the angular speed of the pressure variation, t the time and φ the phase shift. Figure 6.15 shows the idealized pressure signals from three pressure sensors located at different circumferential positions. The signals are spaced out with respect to the annular sensor positions with a repetition of the signal from the sensor at position 0. A line of constant phase of pressure oscillation can be added on the plot. This line links the pressure peak in one signal, corresponding to a stall cell, to the peak in the next signal

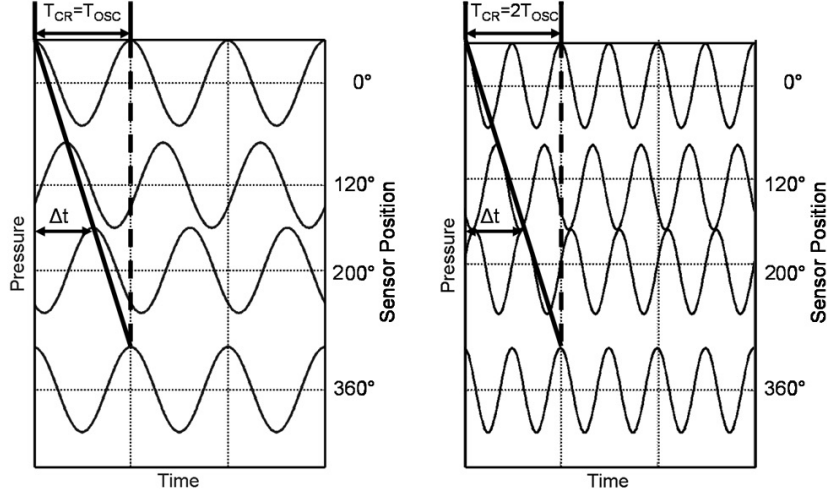


Figure 6.15.: Idealized pressure signals of stall cells from three sensors at different circumferential locations and line of constant phase of pressure oscillation for a single stall cell on the left and a two stall cells case on the right.

corresponding to the same stall cell. The non-equal placement of the sensors around the annulus is essential in order to distinguish the stall cells in case the number of cells is a multiple of the number of sensors. Following this principle, the line of constant phase follows the same stall cell through the different sensors. The time between the crossing of the constant phase line and the pressure signal of the sensor at position 0 and 360° is the time for the complete rotation of one stall cell t_{CR} . The time of one pressure oscillation t_{OSC} is defined as the time between two maxima of the signal. Hence, the number of stall cells n can be calculated by dividing the time of a cell rotation by the time of a pressure oscillation:

$$n = \frac{t_{CR}}{t_{OSC}} \quad (6.12)$$

In the examples presented on Fig.6.15 the number of stall cells determined with this method is 1 for the example on the left and 2 for the example on the right.

This method is applied to the three experiments presented below in order to determine the number of stall cells during the various parts of the signals. Figure 6.16 shows the first oscillations of the pressure signals from the three fast sensors after the stall inception in experiment 1. The signal of sensor $K7$ is repeated at

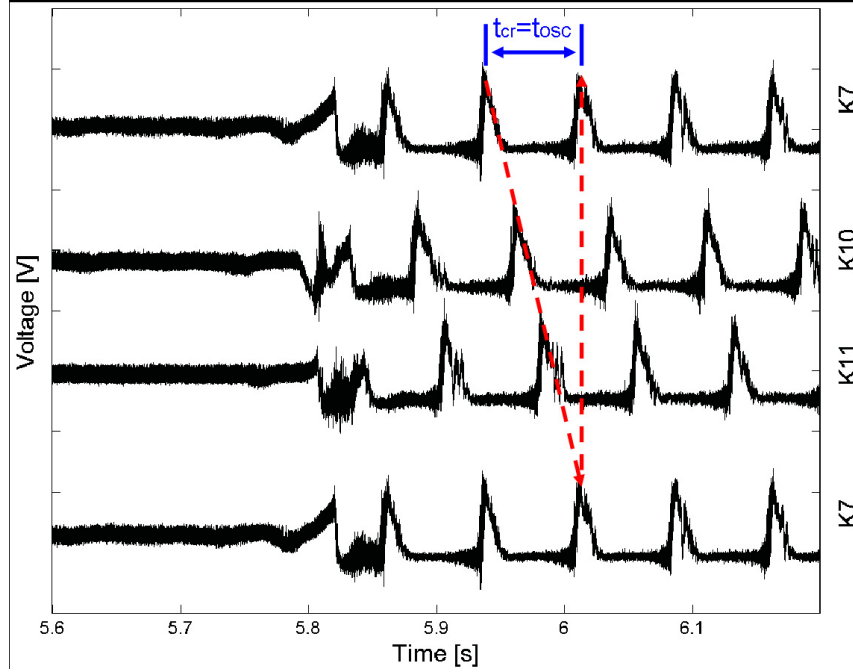


Figure 6.16.: Part *II* of the fast sensors voltage signals in experiment 1.

the bottom to show the behavior over the full circumference. The arrangement of the signals along the y axis is proportional to the angular location of the sensors in the casing wall. The red line corresponds to the constant phase line as described above. The application of the method of Levy [70] allows determining that the number of stall cells of the low frequency part (part *II*) of the first experiment is:

$$n = \frac{t_{CR}}{t_{OSC}} = 1 \quad (6.13)$$

The pressure signal from the three sensors around the point of frequency shift in experiment 1 are presented on Fig.6.17. On the left hand side, a single stall cell configuration is visible. On the right hand side, after the frequency shift occurred, the number of stall cells is:

$$n = \frac{t_{CR}}{t_{OSC}} = 2 \quad (6.14)$$

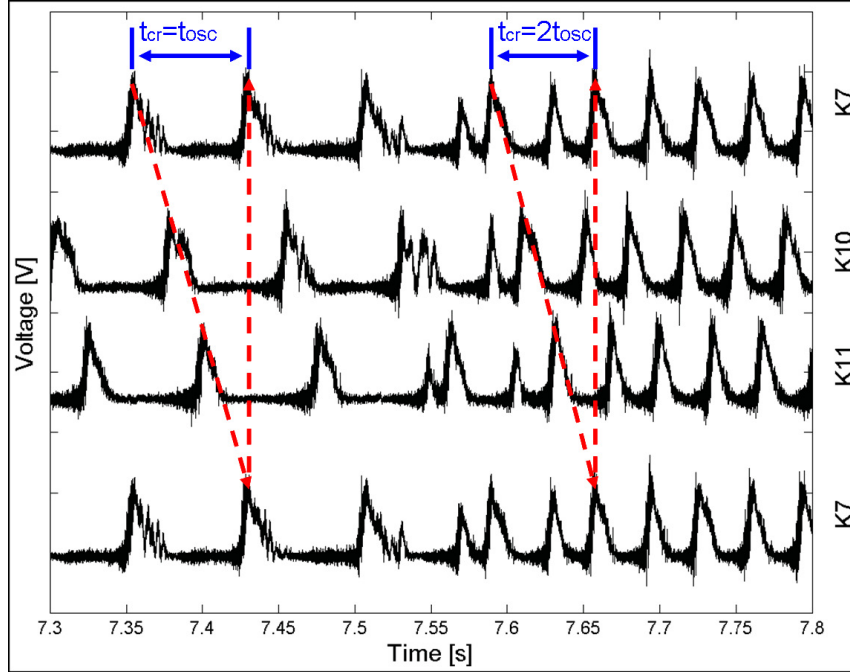


Figure 6.17.: Transition between parts *II* and *III* of the fast sensors voltage signals in experiment 1.

This bears out that the frequency variation appearing in the pressure signals for rotating stall regime is due to a change of stall cells number. In the previous section it was shown that this change is caused by a cell splitting. In this experiment, the compressor passes from a regime with a single stall cell to a stable two stall cell configuration. This process and the number of cells during the various parts of the signal are identical in experiment 2 as in experiment 1, except for the time during the stall inception and the cell splitting, which is shorter.

A zoom on the frequency shift point of the pressure signals in experiment 3 is presented on Fig.6.18. In this case the splitting of the stall cell leads the compressor from a single cell configuration to a regime with three stall cells.

Once the number of stall cells n and the frequency of pressure oscillation f_{OSC} is determined, the frequency of cell rotation f_{CR} is calculated as :

$$f_{CR} = \frac{f_{OSC}}{n} \quad (6.15)$$

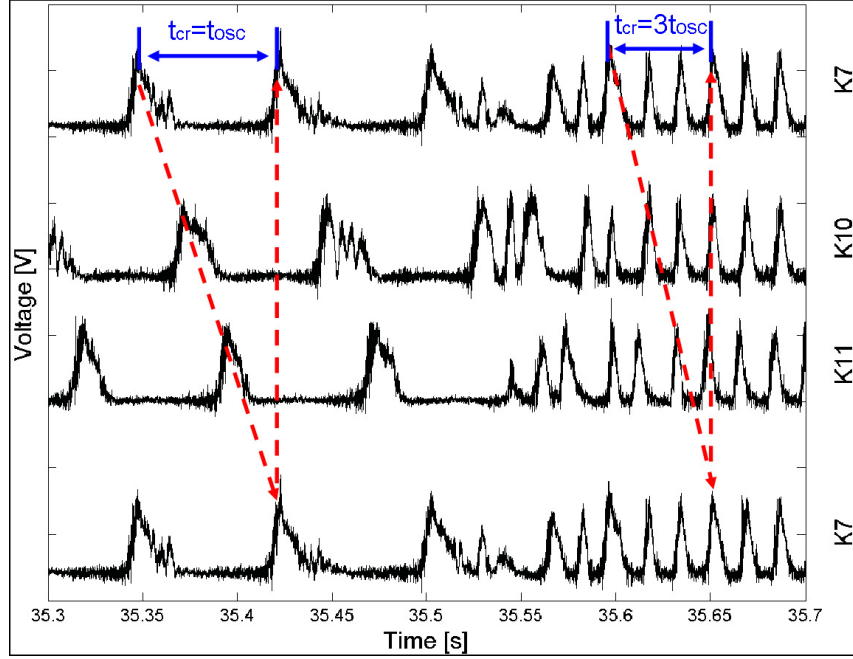


Figure 6.18.: Transition between parts *II* and *III* of the fast sensors voltage signals in experiment 3.

The frequency analysis based on the wavelet map and the FFT in experiment 1 shows a frequency of 13.4 Hz during part *II* of the signal. The number of stall cells during this period is one. It means that the stall cell is rotating with a frequency of 13.4 Hz . The mean rotational speed of the rotor during rotating stall corresponds to a frequency of 79.7 Hz . Thus the rotational speed of the cell is 16.8% of the rotor rotational speed. After the splitting into two cells, the frequency of the rotating stall is 29.4 Hz , giving a rotational frequency of one cell of :

$$f_{CR} = \frac{29.4 \text{ Hz}}{2} = 14.7 \text{ Hz} \quad (6.16)$$

or 18.4% of the rotor rotational speed.

In experiment 2 the single stall cell regime occurs during such a short time that it is not possible to examine the exact speed of the cell. The frequency during the two cells configuration is also 29.4 Hz , equal to a rotational speed of 18.4% of the rotor rotational speed.

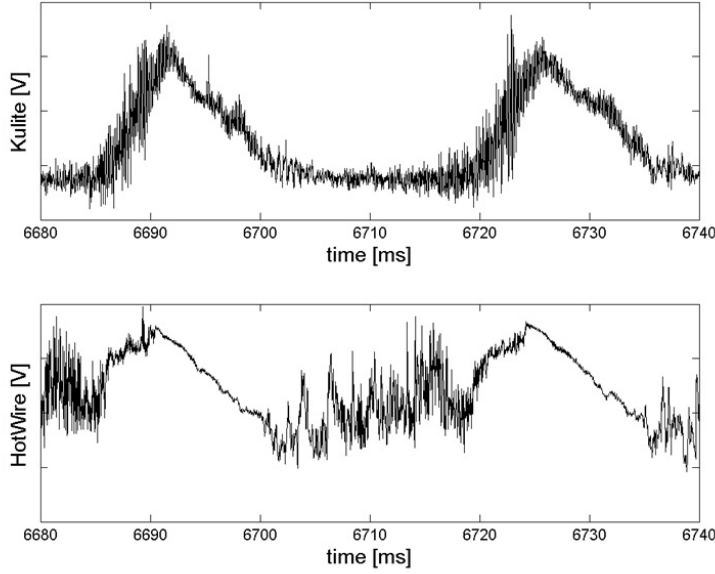


Figure 6.19.: Simultaneous signal from static pressure sensor and hot wire under stall condition.

At the beginning of experiment 3 a single stall cell is rotating at 13.4 Hz . After the splitting into three stall cells, the frequency of the stall phenomenon is 57.1 Hz . This leads to a stall cell rotational speed of respectively 16.8% and 23.9% of the rotor rotational speed.

Once the number and the speed of the rotating stall cells is known, the extension of these cells has to be determined. The voltage signals from a static pressure sensor and a hot wire probe corresponding to a stall cell are presented on Fig.6.19. As explained before, the calibration curve of the pressure sensors are reversed. This means that a low pressure voltage corresponds to a high static pressure, and thus to a low velocity region. The region of low pressure voltage, limited by two peak of high voltage, corresponding to high velocity, is the stall cell. Indeed, we can observe on the figure that the region presenting a low voltage in the pressure signal, presents at the same time a high turbulence in the velocity signal. . This confirms that this region corresponds to the extension of the stall cell.

The extension of the stall cell is defined as the part of the signal where the static pressure is higher than the mean value of the static pressure over the complete analyzed signal. The percentage of circumference covered by one stall cell $\phi_{covered}$ is calculated by multiplying the duration of the high value in the signal Δt_{Cell} by

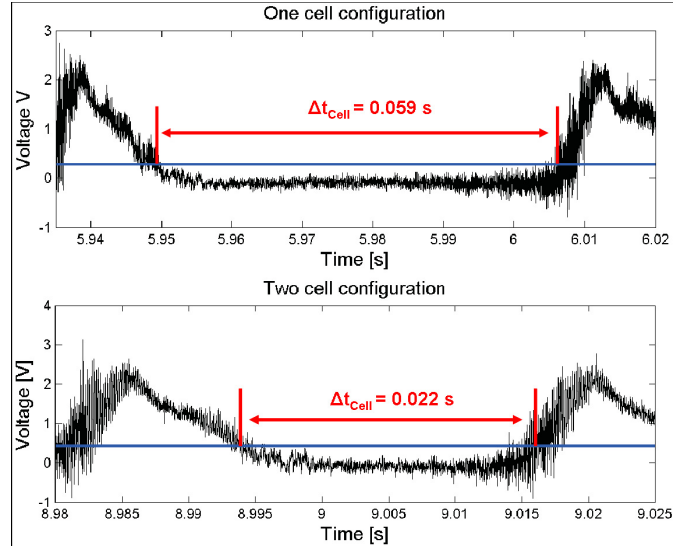


Figure 6.20.: Determination of the stall cell extension based on graphical observation in experiment 1.

the frequency of cell rotation:

$$\phi_{\text{covered}} = \Delta t_{\text{Cell}} * f_{\text{CR}} * 100 \quad (6.17)$$

Fig.6.20 presents a zoom on one stall cell in the configuration with one (above) and two (below) cells in experiment 1. These plots allow determining the duration of the stall cell Δt_{Cell} . Based on this duration the circumference covered by the stall cell in the single cell configuration is 79%. In the two cells configuration the circumference covered by one stall cell is 32.3% and the total circumference covered by all cells is 64.6%.

The size of the stall cells in experiment 2 and 3 is determined by means of Fig.6.21 and 6.22. The observations give in experiment 2, in a two stall cells configuration, a circumference covered by one cell of 32.3% (64.6% for the two cells). The circumference covered in experiment 3 is 80.4% for the single cell regime, and, for the three cells regime, 20.2% for one cell and 60.5% for all cells.

Several experiments have been performed for the casing treatment application as well. Similar results are obtained in terms of number, rotational speed and size of stall cells. The casing treatment does not influence at all the unstable regime of this compressor. This is easily explained by the fact that this compressor is hub critical and that the unstable regime is initiated by the separation area at the hub

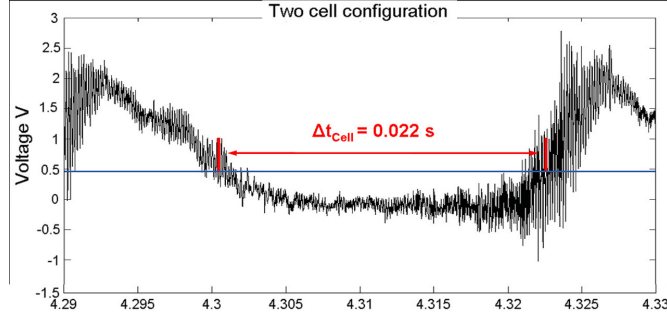


Figure 6.21.: Determination of the stall cell extension based on graphical observation in experiment 2.

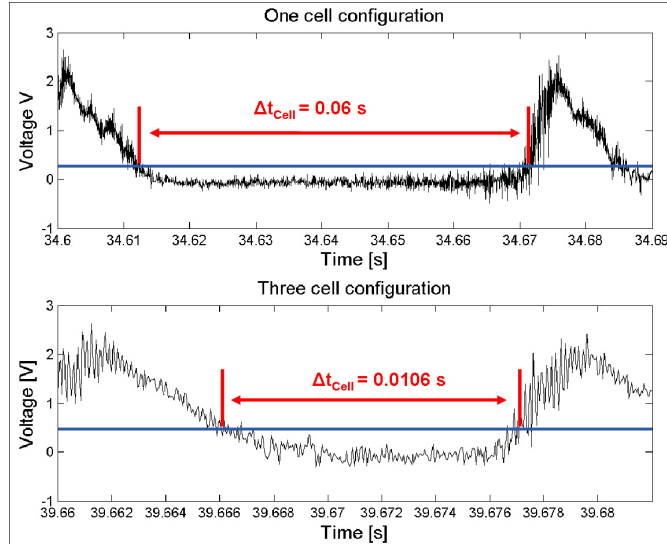


Figure 6.22.: Determination of the stall cell extension based on graphical observation in experiment 3.

part of the rotor, as shown in sections 4.3 and 5.3. A change of the tip flow by the use of casing treatment has no influence on this separation process and thus on the unstable regime.

In part *III* of the signals, the compressor presents an unstable configuration with multiple full-span stall cells. According to Day [26], this is an unusual configuration. Furthermore, Day declared that in rare cases where multiple full-span

stall cells are present, these cells are not necessarily equally-spaced or of equal size. The results presented in this thesis show a multiple full-span stall cells configuration with equally-spaced cells of equal size.

6.4. Conclusion

In the present chapter, we have presented a complete experimental characterization of the rotating stall occurring in the compressor. The study is based on the measurements performed with three fast pressure sensors and four hot-wires set up in a special arrangement. The fast sensors are put in the same axial position and un-evenly spread around the annulus. They are used for the determination of the type of the unsteady phenomena and their inception processes, as well as the characteristics of stall cells. The hot-wires are placed in the same circumferential position in the four axial measurements planes. This arrangement allows the specification of the stall cells extension over the stage.

The spatial arrangement of the fast pressure sensors and an adequate plotting of the pressure signals allow a visualization of the phase shift of the unsteady phenomena between the various sensors. This shift indicates a rotation of the phenomena around the annulus, corresponding to a rotational stall and not to surge. On the pressure signals, low frequency oscillations are visible about twenty rotor revolutions before the stall inception point, suggesting a modal wave stall inception process. The spatial extension of the stall cells is determined using the hot-wire measurements. They indicate full-span stall cells covering axially the entire stage from the inception point.

An evolution over time of the shape of the rotating stall phenomena is observed in the pressure signals. This evolution is studied in the frequency domain using the Fourier transformation and the wavelet transformation of the signals. The evolution is characterized by a change in the frequency of the unsteady phenomena and is due to a splitting of the stall cells. A difference in the timing of this evolution and the frequency change is observed between three tests performed under the same conditions.

A characterization of the stall cells of three experiments performed in the same conditions is detailed. This characterization concerns the number and the rotational speed of the stall cells, as well as the circumference of the rotor covered by these cells. Table 6.1 below gives the results for these three different experiments as well as for one experiment performed in the casing treatment configuration.

According to Day [26], the stall configuration observed in this compressor is completely unusual. The results show a stall configuration with a single full-span stall cell splitting in multiple full-span stall cells with equally-spaced cells of equal size, rotating at maximum 24% of the rotor rotational speed. Firstly, Day observes that full-span cells are characteristic of high pressure compressors. In this case, it is clearly a low pressure compressor. Two elements can explain the presence of full-span stall cells. The first is the very short blade, due to the contra-rotating

Casing config.	Nr. Experiment	Number of stall cells	Cell speed [%]	Covered circumference of one cell [%]	Covered circumference of all cells [%]
Smooth	1	1	16.8	79	79
		2	18.4	32.3	64.6
	2	1	-	-	-
		2	18.4	32.3	64.6
	3	1	16.8	80	80
		3	23.9	20.2	60.5
CT	1	1	16.8	79	79
		2	18.4	33.8	67.6

Table 6.1.: Summary of the stall cells characteristics in the compressor stage during tests presenting various behavior, and for the smooth casing and casing treatment configurations

architecture and the scaling of the test section. The second element is that the origin of the blade stall is the flow separation located at the hub. And then the stall extends radially to the tip, covering all the blade height. Secondly, Day observed that full-span cells are very seldom multiple. And, in the rare cases of multiple full-span cells, they are not equally spaced or of equal size. And they generally merge into a single cell. Here, we observe the opposite behavior. A single full-span cell splits into multiple full-span cells, equally spaced and of equal size. Finally, the speed of the stall cells observed in this compressor are out of the range described by Day [27]. In his presentation, Day observed a single stall cell covering 80% of the circumference and rotating at 32% of the rotor speed. In our case, the single stall cell covers the same proportion of the circumference but rotates at the half of that speed. Nevertheless, this very low speed of full-span stall cell has been observed by Pampreen [85] in the case of high design flow coefficient. The very large circumference covered by the stall cells induces that the compressor has to be turned off to escape the stall.

The circumferential groove casing treatment applied to the compressor has no influence on the rotating stall inception process as well as on the characteristics of the stall phenomena. This is due to the fact that this compressor’s instability comes from the separation zone at the hub, induced by the hub corner vortex. And this vortex is not influenced by a modification of the tip flow due to a casing treatment.

Chapter 7.

Conclusions and Perspectives

7.1. Conclusions

This work presents the results of a complete investigation of a single stage low pressure compressor. This compressor is representative of the first stage of a booster typical of the new contra-rotating turbofan and characterized by unusually high load- and flow coefficient, and low aspect ratio. The number of references concerning the low-pressure compressors is very limited. And, to date, there is no literature about the low-pressure compressors of contra-rotating turbofan architecture. This thesis represents thus a reference basis about this topic.

The overall performance of the compressor as well as of the rotor alone is presented for various operating conditions from choke to near stall. The study of the compressor stage has been completed at two Reynolds numbers, corresponding to the cruise and the take-off conditions. A description of the three-dimensional unsteady secondary flows present in the rotor, as well as their impact on the flow blockage in the rotor row is given in this thesis.

The phenomena of instability occurring in the compressor stage are presented in terms of origin, inception mode, unsteady behavior and geometrical characteristics. The type of instability observed in the compressor is related to various indications in the performance of the stage.

The effects of a circumferential groove casing treatment on the overall performance and on the stability of the compressor stage, as well as on the unsteady flow field in the rotor, have been described.

The experimental results are compared to numerical simulations performed with various codes and turbulence models. This comparison has been made on the overall performance of the stage and of the rotor, as well as on the effect of the casing treatment and the Reynolds number. This allows validating the use of the CFD in the design of this type of unconventional compressors and of casing treatments.

Compressor Performance and Flow Characteristics The low-pressure compressor considered in this work has been designed for the new contra-rotating turbofan architecture. The determination of its global performance and of the phenomena occurring in the compressor is fundamental. Various instruments and special post-processing procedures were used in the frame of this work. Due to the very small

size of the blades, a special set of steady and unsteady miniaturized probes was used. The data processing programs were developed for this application, including the phase lock averaging and the interpolation methods.

The global performance is derived from the measurement of the conditions upstream and downstream of the stage. Upstream, the turbulence level is between 1.25% near the hub up to 2.5% at 80% of the blade height. Above 80%, the turbulence level is highly influenced by the instrumentation. The radial distribution of the total pressure at the inlet of the stage presents a displacement thickness at tip seven times higher than at hub, due to the inlet geometry of the compressor. Downstream the compressor, a large low pressure region along the suction side of the blade is visible. This is caused by the corner vortex secondary flow. Under cruise conditions, the reduced mass flow of this compressor is 0.086 at choke, 0.077 at design, and 0.053 near stall. The pressure ratio of this compressor stage goes from 1.101 at choke and near stall to 1.127 at design, for an efficiency of 67.7% at choke, 80.8% at design, and 73.8 near stall. The stall margin, based on the reduction of the reduced mass flow and the pressure ratio, is about 30%. The total-to-total pressure ratio as a function of the mass flow shows a peak at design. The reduction of the pressure ratio near stall is caused by the development of a pressure defect region due to the hub corner stall. This performance trend is characteristic of the hub loaded compressors.

The performance of the compressor has been investigated under two pressure conditions, representing the *cruise* and *take – off* conditions. This allows investigating the effect of the Reynolds number on the performance and the flow characteristics. The measurement at high Reynolds number shows reduced mass flows passing through the stage at the various working points identical as under cruise conditions. In terms of pressure ratio and efficiency, a higher Reynolds number leads to an increase of the performance. This is explained by the stabilization effect on the boundary layer by the increase of the Reynolds number.

The investigation of the unsteady phenomena occurring in the rotor required specific instrumentation characterized by a fast response and a high level of miniaturization. The unsteady casing wall static pressure and the total pressure downstream the rotor were acquired simultaneously for the various working and pressure conditions. In the casing, the measurements were performed by the means of inserts placed between 45% upstream the leading edge and 45% downstream the trailing edge. A phase lock averaging procedure, based on the one-per-revolution trigger, was developed and applied to the unsteady pressure signals in order to determine the mean pressure distribution over one pitch and the fluctuation around this mean value (RMS). Then, the pressure traces at the different positions were used to interpolate the pressure fields. Due to the low resolution in the axial direction, the interpolation direction has a very important influence on the result of the interpolation. After investigation, a specific direction was imposed for the different region of the measurement in order to match the flow. This procedure was applied to both the unsteady casing wall static pressure and the total pressure downstream the rotor.

The use of the simultaneously acquired unsteady casing static pressure together with the rotor exit pressure gives a very good visualization in three dimensions of the tip leakage vortex and allows determining its origin and extension path. From choke to near stall, the origin of the vortex is moving towards the leading edge and the slope of the vortex axis is increasing, as already observed in several investigations on this field.

The unsteady total pressure field downstream the rotor shows a large low pressure area located along the suction side of the blade in the hub region. This is due to a flow separation on the suction side of the blade. Near stall, this separation area covers 35% of the pitch. This very high source of blockage and loss is considered as the origin of the instability in the compressor.

Instability Phenomena The reduction of the mass flow passing through a compressor can create instability phenomena called rotating stall and surge. In the case of the compressors of an aircraft engine, the back flow resulting from these phenomena can lead to the destruction of the compressors by the passage of hot gas from the combustion chamber. A complete comprehension of the phenomena of stall and surge occurring in a compressor is fundamental for safety reasons. The stall and stall inception process occurring in this compressor have been completely characterized. The evolution of the unstable regime was described by means of the wavelet analysis, allowing the observation of a phenomenon in the temporal and frequency domain together.

Measurements of unsteady static casing wall pressures at various circumferential positions show a rotating stall as unstable regime. The unsteady pressure traces show that the stability limit is preceded by long wavelength disturbances, corresponding to the typical modal stall inception process. This inception mode is confirmed by the fact that the stability limit of the compressor is reached at or close to zero-slope of total-to-static characteristic [106]. Furthermore, the fact that the trajectory of the tip leakage vortex close to the stability limit remains in the blade passage is another criterion for modal stall inception. The inception mode can be linked to the fact that, in this highly hub loaded compressor, the stall is caused by a large separation area at the hub of the blades. The presence of a blade stall is supported by the determination of the Greitzer criterion [43].

The rotating stall observed in the compressor is characterized by full-span stall cells extending axially over the complete stage. Starting with a single stall cell, the stall configuration evolves in time by the splitting of the cell in two, three or four cells. The stall cells configuration observed in the compressor is completely unusual. According to Day [26], the full-span cell, characteristics of a high pressure compressor, is single. In the rare cases of multiple full-span cells, they are not equally spaced or of equal size. And they generally merged into a single cell. In this case, one single full-span cell splits into multiple full-span equally spaced cells of equal size. The very low speed of the single stall cell, 17% of the rotor speed for 80% of the circumference covered, was already pointed by Pampreen [85] in the

case of high design flow coefficient.

Casing Treatment Effects The effect of a 3x1 mm circumferential groove casing treatment, located at 15 % of the axial chord, on the compressor performance and on the flow field at the rotor tip has been investigated. This effect is negligible on the performance calculated at mid-span, meaning that the casing treatment does not provide an improvement of the stability margin of the compressor. This confirms the non effectiveness of casing treatments on compressors presenting a blade stall observed by Greitzer [43] and Houghton [54], and is caused by the fact that this compressor is hub critical.

Nevertheless, an increase of the mass flow and of the pressure ratio due to the casing treatment is observed at the stability limit operating point on the performance calculated from the complete mapping at the outlet of the stage. This is caused by a reduction of the blockage created by the tip leakage vortex in the tip region. This is showed by the mapping of the casing wall static pressure and the total pressure downstream the rotor. The reduction of the blockage corresponds to a reduction of the main flow/tip vortex interaction represented by the RMS amplitude. The presence of the circumferential groove leads to a circumferential displacement of the tip leakage vortex core as well. These two effects of the casing treatment on the flow field are only visible close to the stability limit, when the origin of the vortex is located upstream the casing treatment.

CFD Confidence The experimental results have been compared to numerical simulations performed with two CFD codes, FINE and elsA. In the case of the elsA code, three different turbulence models have been used, $k-\epsilon$, $k-\omega$ and $k-l$. The inlet and outlet conditions imposed for the simulations were the values measured in the compressor stage.

The simulated global performance shows a relatively good agreement with the experiments. Nevertheless, the CFD codes do not allow simulating conditions close to the stability limit due to the fact that at these conditions the flow is mainly driven by unsteady phenomena which are hard to capture with steady RANS simulations. Furthermore, an over-estimation of the pressure ratio and an under-estimation of the efficiency is caused by the difference between the geometry of the real test section and the ideal domain considered by the CFD. The results from the three turbulence models used with the elsA code present discrepancies larger than the experimental uncertainty. The $k-\epsilon$ model presents the best compromise in terms of performances predictions.

The simulation of the performance of the rotor alone and the unsteady flow field around the rotor fits very well with the experiments. The tip leakage vortex extension and behavior is very well captured by the CFD. The development of the hub corner stall is very well simulated as well.

The two codes used for CFD calculation running with the same turbulence model show opposite influence of the casing treatment on the global performance of the

compressor. Furthermore, this influence is lower than the difference between the performance calculated by the three turbulence models with the elsA code. This, and the very important discrepancy between the performance calculated with the same code and various turbulence models, mean that great care is required when making decisions based on CFD.

7.2. Perspectives

This work presents a large part of the characteristics of an unusual design of a low pressure compressor, including the performance, unsteady flow structure and unstable regime. Nevertheless, additional works are needed to have a full understanding of this compressor and the related phenomena. These additional works concern mainly the method used for the interpolation of unsteady casing pressure traces, the acquisition of the unsteady pressures, and the determination of the efficiency.

The efficiency of the compressor considered in this thesis is the isentropic efficiency based on flow characteristics measured at the inlet and the outlet of the stage. Another method for the determination of the efficiency is very often used in the industry. This method is the determination of the mechanical efficiency by measuring the work passing through the shaft and the work effectively performed by the compressor. The difference between the isentropic and the mechanical efficiencies is the loss in the bearings and the aerodynamic loss around the rotating disk in the core of the compressor. These losses can be determined by measuring the work through the shaft when the compressor is rotating under vacuum for the bearing loss, and when the compressor is running under design pressure conditions without the compressor blades for the aerodynamic loss due to the rotating disk.

The investigation of the tip leakage flow structure and the stall is based on the measurement of the unsteady casing static wall pressure. In order to protect the expensive unsteady pressure sensors and to limit the sensing area of these sensors, the sensors are placed in an insert. This configuration leads to the presence of a cavity in front of the sensor. The exact effect of this cavity on the measurement in terms of phase and signal amplitude is currently not fully known and should be investigated in detail.

The stall phenomena occurring in the compressor are investigated by means of only three unsteady pressure sensors located in the casing while eleven were available. This comes from the limitation of acquisition power and the necessity of long acquisition times at high frequency for the stall investigation. Nevertheless, the acquisition of all the unsteady pressure sensors, covering 190% of the blade chord, during the passage from stable to unstable regime could give important information about the development of the stall cells.

The effect of the IGV and stator on the unsteady pressure measurement in the casing and downstream the rotor have been completely neglected. But this effect can be non-negligible, especially if the sensor for the measurement of the

casing static pressure is located in the wake of the IGV. The investigation of the importance of this effect should be performed by measuring the unsteady pressure field for various stators positions.

The interpolation method used for the determination of the unsteady pressure field around the rotor should be improved. Indeed, if the interpolation method developed and used in this work gives satisfactory results, the choice of the interpolation direction is manual. This means a human interpretation of the results that can be wrong, leading to a wrong pressure field. An automatic procedure should be developed to skip the human intervention. This new procedure could be based on the Essentially Non-Oscillatory (ENO) or Weighted Essentially Non-Oscillatory (WENO) schemes [47] [59] [77]. These finite difference schemes, based on the adaptation of stencils in order to achieve high order accuracy, and on a non-oscillating property near discontinuities, give very good results for the interpolation of complex structures. Finally, the determination of the direction of the interpolation should be based on the RMS traces. Indeed these traces give a more accurate location of the various phenomena present in the flow.

Finally, a last point that requires some additional work and investigation is the numerical simulations. Indeed, the results of the computation with various codes and turbulence models show very high discrepancies. The importance of the influence of the turbulence model on the results has been highlighted by using various models with the same solver and the same mesh. The investigation on this influence should be completed by a comparison of the entropy maps in order to determine the origin of the difference between the models. Also, the influence of the geometry, the mesh, or the solver has not been investigated in this thesis. This should be done in a future numerical work.

Appendix A.

Uncertainty Calculations

A.1. Thermocouple probe uncertainty

$$\frac{\partial T_0}{\partial M} = \frac{-T_J \cdot (1-r) \cdot A'}{(1-(1-r) \cdot A)}, \quad (\text{A.1})$$

where A is the term including the Mach number:

$$A = \frac{(\gamma - 1) M^2}{2 + (\gamma - 1) M^2}. \quad (\text{A.2})$$

The derivation of this term in function of the Mach number gives:

$$\begin{aligned} A' &= \frac{2 \cdot (\gamma - 1) \cdot M \cdot (2 + (\gamma - 1) M^2) - (\gamma - 1) \cdot M^2 \cdot 2 \cdot (\gamma - 1) \cdot M}{(2 + (\gamma - 1) M^2)^2}, \\ &= \frac{4 \cdot M \cdot (\gamma - 1)}{(2 + (\gamma - 1) M^2)^2}, \end{aligned} \quad (\text{A.3})$$

The equation A.1 can thus be written as:

$$\frac{\partial T_0}{\partial M} = \frac{-T_J \cdot (1-r) \cdot 4 \cdot M \cdot (\gamma - 1)}{\left[\left(1 - (1-r) \frac{(\gamma-1)M^2}{2+(\gamma-1)M^2} \right) \cdot ((2 + (\gamma - 1) M^2)) \right]^2} \quad (\text{A.4})$$

This equation can be simplified by:

$$\begin{aligned} \frac{\partial T_0}{\partial M} &= \frac{-T_J \cdot (1-r) \cdot 4 \cdot M \cdot (\gamma - 1)}{[2 + (\gamma - 1) M^2 - (1-r) (\gamma - 1) M^2]^2}, \\ &= \frac{-T_J \cdot (1-r) \cdot 4 \cdot M \cdot (\gamma - 1)}{[2 + r (\gamma - 1) M^2]^2} \end{aligned} \quad (\text{A.5})$$

A.2. Three-hole probe uncertainty

Yaw angle uncertainty The determination of the yaw angle is based on the relation between the angle and the K_{yaw} coefficient. As it can be observed on

Fig.4.11, this relation can be considered as linear in a given angular range:

$$\alpha = C_1 K_{yaw} = C_1 \frac{P_L - P_R}{Q} \quad (\text{A.6})$$

where

$$Q = P_c - \frac{P_L + P_R}{2} \quad (\text{A.7})$$

and

$$C_1 = \left(\frac{\Delta\alpha}{\Delta K_{yaw}} \right)_{calib} \quad (\text{A.8})$$

From this relation, the uncertainty on the flow angle can be calculated using the equation:

$$\delta\alpha = \sqrt{\left(\frac{P_L - P_R}{Q} \delta C_1 \right)^2 + \left(\frac{C_1}{Q} \delta(P_L - P_R) \right)^2 + \left(-C_1 \frac{P_L - P_R}{Q^2} \delta Q \right)^2} \quad (\text{A.9})$$

From the definition of the angular sensitivity A.8, the uncertainty on C_1 is determined using the equation:

$$\delta C_1 = \sqrt{\left(\frac{1}{\Delta K_{yaw}} \delta\Delta\alpha \right)^2 + \left(\frac{-\Delta\alpha}{\Delta K_{yaw}^2} \delta\Delta K_{yaw} \right)^2} \quad (\text{A.10})$$

In this equation, the uncertainties $\delta\Delta\alpha$ and $\delta\Delta K_{yaw}$ are respectively twice the uncertainty on the angular positioning $\delta\alpha$ and the calculation of the K_{yaw} during the calibration. The angular positioning uncertainty is equal to 0.01° , hence $\delta\Delta\alpha = 0.02$. The value of the uncertainty on the K_{yaw} , based on its definition 4.24, is given by:

$$\delta K_{yaw} = \sqrt{\left(\frac{1}{Q} \delta(P_L - P_R) \right)^2 + \left(\frac{-(P_L - P_R)}{Q^2} \delta Q \right)^2} \quad (\text{A.11})$$

Finally the uncertainties $\delta(P_L - P_R)$ and δQ are expressed by:

$$\delta(P_L - P_R) = \delta P_L + \delta P_R \quad (\text{A.12})$$

$$\delta Q = \delta P_c + \frac{\delta P_L}{2} + \frac{\delta P_R}{2} \quad (\text{A.13})$$

And the uncertainties on the pressures δP_c , δP_L and δP_R are calculated following the method described in section 4.1.1.

Mach number uncertainty The Mach number is function of the flow angle α and the K_{Mach} :

$$M = f(\alpha, K_{Mach}) \quad (A.14)$$

The uncertainty on the Mach number is determined by:

$$\delta M = \sqrt{(\delta M)_\alpha^2 + (\delta M)_{K_{Mach}}^2} \quad (A.15)$$

where $(\delta M)_\alpha$ is the uncertainty on the Mach number due to the uncertainty on the flow angle α and $(\delta M)_{K_{Mach}}$ the uncertainty due to the uncertainty on the K_{Mach} .

The $(\delta M)_\alpha$ is calculated as the maximum variation of the Mach number caused by the variation of the angle α :

$$(\delta M)_\alpha = \max \left\{ \begin{array}{l} abs(f(\alpha, K_{Mach}) - f(\alpha + \delta\alpha, K_{Mach})) \\ abs(f(\alpha, K_{Mach}) - f(\alpha - \delta\alpha, K_{Mach})) \end{array} \right\} \quad (A.16)$$

where $\delta\alpha$ is defined by equation A.9.

The same procedure is used for the calculation of $(\delta M)_{K_{Mach}}$:

$$(\delta M)_{K_{Mach}} = \max \left\{ \begin{array}{l} abs(f(\alpha, K_{Mach}) - f(\alpha, K_{Mach} + \delta K_{Mach})) \\ abs(f(\alpha, K_{Mach}) - f(\alpha, K_{Mach} - \delta K_{Mach})) \end{array} \right\} \quad (A.17)$$

with δK_{Mach} equal to:

$$\delta K_{Mach} = (\delta K_{Mach})_{meas} + (\delta K_{Mach})_{calib} \quad (A.18)$$

with $(\delta K_{Mach})_{meas}$ and $(\delta K_{Mach})_{calib}$ given, from equation 4.27, by:

$$\delta K_{Mach} = \sqrt{\left(\frac{P_L + P_R}{2} \delta P_c\right)^2 + \left(\frac{-0.5 P_c}{Q^2} \delta P_L\right)^2 + \left(\frac{-0.5 P_c}{Q^2} \delta P_R\right)^2} \quad (A.19)$$

where P_c , P_L and P_R are the pressures measured during the static calibration for $(\delta K_{Mach})_{calib}$ and during the test campaigns for $(\delta K_{Mach})_{meas}$, and δP_c , δP_L and δP_R their uncertainties as defined in section 4.1.1.

Total pressure uncertainty From equation 4.25, the total pressure can be expressed by:

$$P_t = K_{Tot} \cdot \left(P_c - \frac{P_L + P_R}{2} \right) + P_c \quad (A.20)$$

Then the uncertainty on the total pressure is given by:

$$\delta P_t = \sqrt{(Q \delta K_{Tot})^2 + ((K_{Tot} + 1) \delta P_c)^2 + \left(\frac{-K_{Tot}}{2} \delta P_L\right)^2 + \left(\frac{K_{Tot}}{2} \delta P_R\right)^2} \quad (A.21)$$

The uncertainty δK_{Tot} is divided in two terms, corresponding to the uncertainty ascribable to the uncertainty on the flow angle α , $\delta K_{Tot,alpha}$, and to the uncertainty on the Mach number M , $\delta K_{Tot,Mach}$. These two terms are defined as the maximum variation of K_{Tot} due to the flow angle or Mach number uncertainty. The calculation of these terms is done following the same procedure as for the $(\delta M)_\alpha$ presented in the previous paragraph. And δK_{Tot} is given by:

$$\delta K_{Tot} = \sqrt{(\delta K_{Tot,alpha})^2 + (\delta K_{Tot,Mach})^2} \quad (\text{A.22})$$

Static pressure uncertainty The static pressure can be expressed by:

$$P_s = P_t - K_{Dyn} \cdot \left(P_c - \frac{P_L - P_R}{2} \right) \quad (\text{A.23})$$

and its uncertainty by:

$$\delta P_s = \sqrt{(\delta P_t)^2 + (Q \delta K_{Dyn})^2 + (K_{Dyn} \delta P_c)^2 + \left(\frac{K_{Dyn}}{2} \delta P_L \right)^2 + \left(\frac{K_{Dyn}}{2} \delta P_R \right)^2} \quad (\text{A.24})$$

The uncertainty on δK_{Dyn} is determined using the same approach as for the uncertainty on δK_{Tot} set out in the previous paragraph.

Appendix B.

Unsteady casing wall static pressure fields at choke conditions

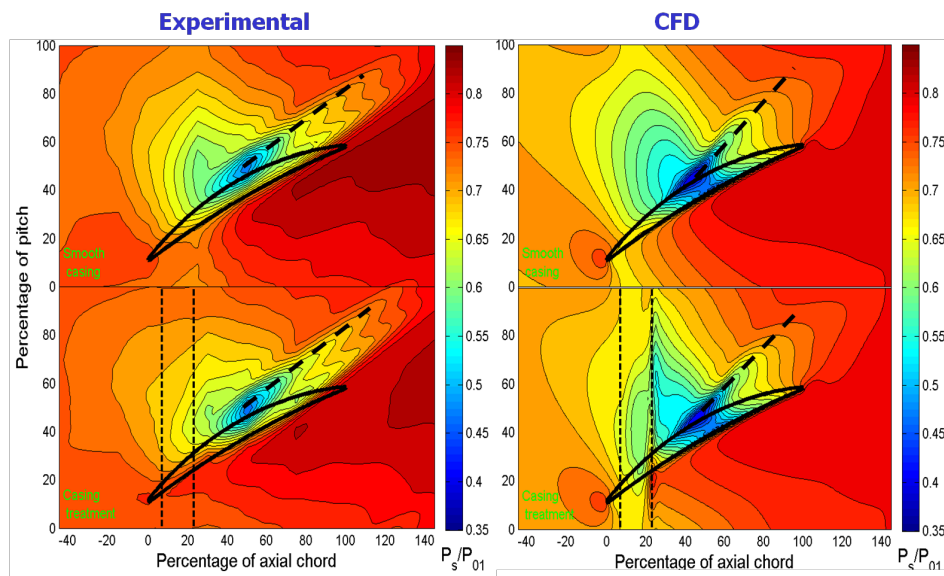


Figure B.1.: Experimental (left) and numerical (right) casing static pressure field for smooth casing (top of the figures) and casing treatment configuration (bottom of the figures) under choke working conditions (point *C*)

186 Chapter B. Unsteady casing wall static pressure fields at choke conditions

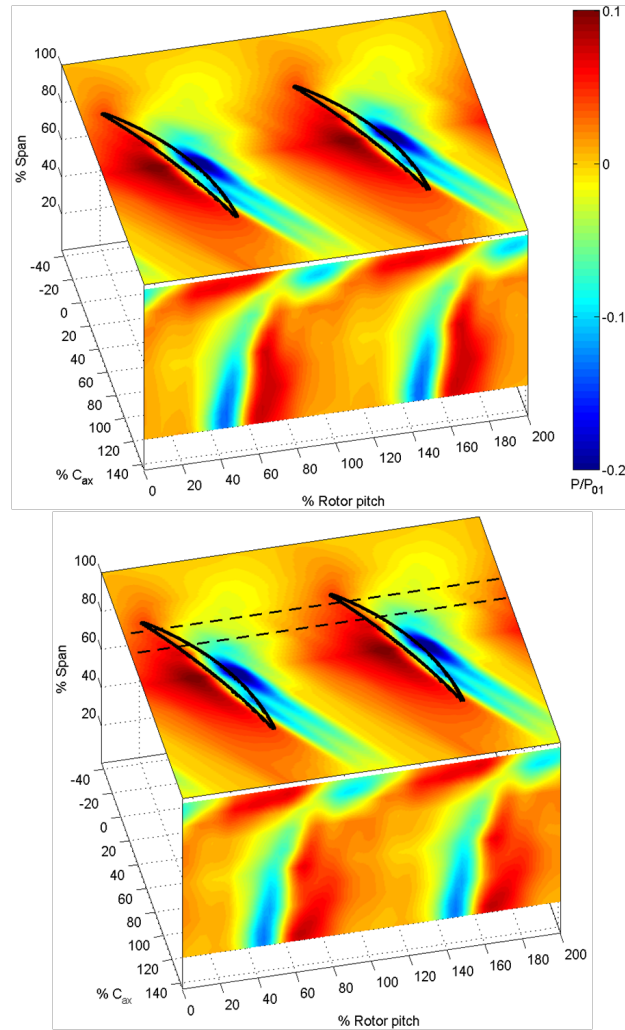


Figure B.2.: Simultaneous visualization of the experimental pressure field in the planes of the casing and downstream the rotor, under choke working conditions for a smooth casing (top) and casing treatment (bottom) configurations

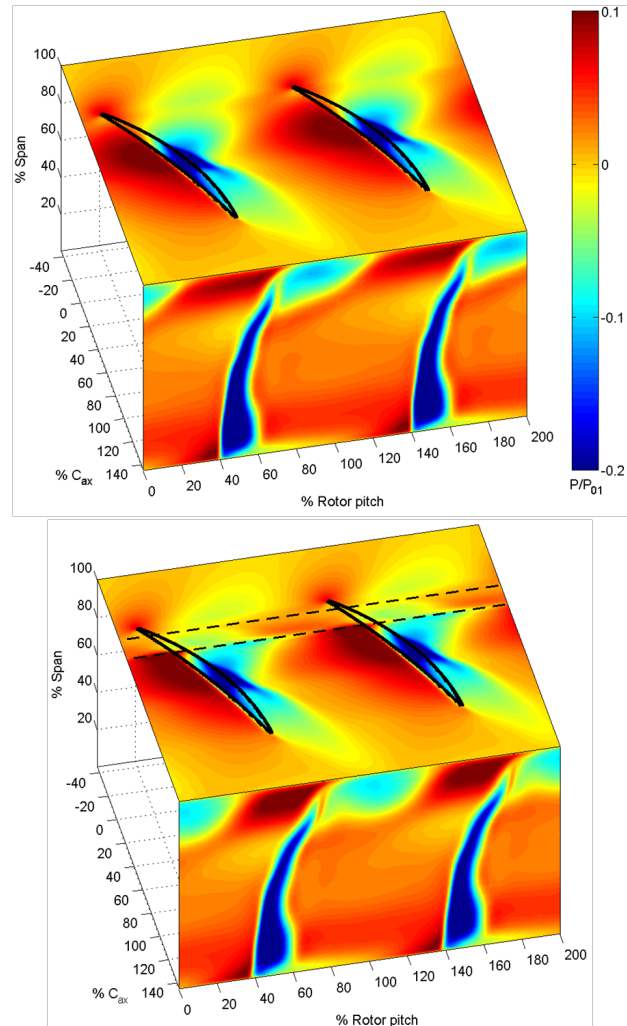
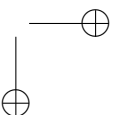
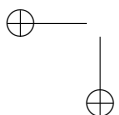
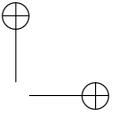
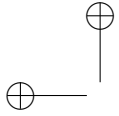


Figure B.3.: Simultaneous visualization of the numerical pressure field in the planes of the casing and downstream the rotor, under choke working conditions for a smooth casing (top) and casing treatment (bottom) configurations



Bibliography

Each reference is followed by the list of the pages where it is cited.

- [1] Measurement of gas flow by means of critical flow venturi nozzles, 1990. 45
- [2] *Sixth Framework Programme Priority 4 - Aeronautics and Space - Annex 1 - "Description of Work"*, 2004. 2, 3, 201
- [3] E. Andrade. Flow uniformity and measurements uncertainties for the axial compressor in the r4 facility (vital project). Master's thesis, von Karman Institute for Fluid Dynamics, 2007. 50
- [4] T. Arts, H. Boerrigter, J.-M. Buchlin, M. Carbonaro, G. Degrez, R. Denos, D. Fletcher, D. Olivari, M. L. Riethmuller, and R. A. Van de Braembussche. *Measurement techniques in fluid dynamics - An introduction*. von Karman Institute for Fluid Dynamics, 2001. 60
- [5] E. E. Bailey. Effect of grooved casing treatment on the flow range capability of a single-stage axial-flow compressor. Technical report, Technical Report TM X-2459, NASA, 1972. 34, 35
- [6] E. E. Bailey and C. H. Voit. Some observations of effects of porous casing on operating range of a single axial-flow compressor rotor. Technical report, Technical Report TM X2120, NASA, 1970. 33
- [7] B. S. Baldwin and H. Lomax. Thin layer approximation and algebraic model for separated turbulent flow. In *AIAA Paper 78-257*, 1978. 52
- [8] B. H. Beheshti, P. C. Teixeira, J. A. Ivey, K. Ghorbanian, and B. Farhanieh. Parametric study of tip clearance casing treatment on performances and stability of a transonic axial compressor. *Journal of Turbomachinery*, 126:527–535, 2004. 37
- [9] J. Bergner, M. Kinzel, and H. Schiffer. Short to length-scale rotating stall inception in transonic axial compressor - experimental investigation. In *Proceedings of ASME Turbo Expo 2005: Power for Land, Sea and Air*, number GT 2006-90209, 2006. 17, 18, 30, 201
- [10] G. A. Brignole, F. Danner, and H.-P. Kau. Time resolved simulation and experimental validation of the flow in axial slot casing treatment for transonic axial compressor. In *Proceedings of ASME Turbo Expo 2008: Power for Land, Sea and Air*, 2008. 36

- [11] J.-F. Brouckaert. *Development of fast response aerodynamic pressure probes for time-resolved measurements in turbomachines*. PhD thesis, von Karman Institute for Fluid Dynamics, 2002. 71, 113
- [12] J.-F. Brouckaert, **N. Van de Wyer**, B. Farkas, F. Ullmann, J. Desset, J. de Laborderie, M. Chome, and S. Hiernaux. Unsteady pressure measurements in a single stage low pressure axial compressor: Tip vortex flow and stall inception. In *Proceeding of ASME Turbo Expo 2009: Power for Land, Sea and Air*, 2009. 130, 136
- [13] J.-F. Brouckaert, **N. Van de Wyer**, J. Desset, B. Farkas, J. Gonzales Peres, R. Capart, J.-F. Thomas, N. Issac, and S. Hiernaux. Experimental and numerical investigation of a single stage low pressure compressor for counter-rotating turbofan engine architecture. In *Proceeding of European Turbomachinery Conference 2009*, 2009. 52, 95
- [14] H. H. Bruun. *Hot-Wire Anemometry - Principle and Signal Analysis*. Oxford Science Publication, 1995. 75
- [15] T.R. Camp and I.J. Day. A study of spike and modal stall phenomena in a low-speed axial compressor. *ASME Journal of Turbomachinery*, 120:393–401, 1998. 28, 29, 91, 105, 154
- [16] L. Castillon, G. Billonet, and S. Plot. New abilities of the multiapplication elsa software in the field of turbomachinery flow computation. In *Proceedings of ASME Fluids Engineering Division Summer Meeting*, number FEDSM2002-31095, 2002. 54
- [17] G. T. Chen, C. S. Greitzer, E. M. Tan, and F. E. Marble. Similarity analysis of compressor tip clearance flow structure. *Journal of Turbomachinery*, 113:260–271, 1991. 16, 17
- [18] M. Choi, J. H. Baek, S. H. Oh, and J. Ki. Role of hub-corner-separation on rotating stall in an axial compressor. *Transaction of the Japan Society for Aeronautical and Space Science*, 51:93–100, 2008. 24
- [19] M. Chome. Inlet profiles of the booster stage in the r4 facility (vital project). Master’s thesis, von Karman Institute for Fluid Dynamics, 2007. 81
- [20] M. Chome. Unsteady aerodynamic errors affecting fast response pressure probes. Master’s thesis, Vrije Universiteit Brussel, 2008. 114, 120
- [21] A. J. Crook, E. M. Greitzer, C. S. Tan, and J. J. Adamczyk. Numerical simulation of compressor endwall and casing treatment flow phenomena. *Journal of Turbomachinery*, 115:501–512, 1993. 35
- [22] N. A. Cumpsty. *Compressor aerodynamics*. Longman Scientific and Technical, 1989. 12, 19, 27, 91

- [23] C. Dano. *Evaluation de modeles de turbulence pour la simulation d'écoulements tridimensionnels instationnaires en turbomachines*. PhD thesis, Ecole Centrale de Lyon, 2003. 54
- [24] I. J. Day. Stall inception in axial flow compressor. *Journal of Turbomachinery*, 115:1–9, 1993. 28, 29, 201
- [25] I. J. Day. Axial compressor performance during surge. *Journal of Propulsion and Power*, 10 (3):329–336, 1994. 32
- [26] I. J. Day. The fundamentals of stall and surge - part i: Stall. In *VKI LS 2006-06 on Advances in Axial Compressor Aerodynamics*, 2006. 18, 27, 29, 30, 172, 173, 177
- [27] I. J. Day. Slide of the the fundamentals of stall and surge - part i: Stall. In *VKI LS 2006-06 on Advances in Axial Compressor Aerodynamics*, 2006. 174
- [28] I. J. Day and N. A. Cumpsty. The measurement and interpretation of flow within rotating stall cells in axial compressors. *Journal of Mechanical Engineering Science*, 20:107–114, 1978. 31, 158
- [29] J. D. Denton. Loss mechanism in turbomachines. *Journal of Turbomachinery*, 115:621–651, 1993. 15
- [30] J. D. Denton. Loss mechanism in turbomachines - part ii : Loss generation in turbomachines. In *VKI LS 1999-02 on Turbomachinery Blade Design Systems*, 1999. 14
- [31] Y. Dong, S. J. Gallimore, and H. P. Hodson. Three-dimensional flows and loss reduction in axial compressors. *Journal of Turbomachinery*, 109:354–361, 1987. 20
- [32] R. P. Dring, H. D. Joslyn, and L. W. Hardin. An investigation of axial compressor rotor aerodynamic. *Journal of Engineering for Power*, 104:84–97, 1982. 20
- [33] O. Durmusoglu. Calibration of pneumatic probes, stall detection on an axial compressor. Master's thesis, von Karman Institute for Fluid Dynamics, 2005. 156
- [34] Dantec Dynamics. *Improved Temperature Correction in StreamWare*. Technical note, Dantec Dynamics A/S. 77
- [35] H. W. Emmons, C. F. Pearson, and H. P. Grant. Compressor surge and stall propagation. In *ASME Transaction*, 79:455–469, 1955. 27

- [36] T. Ercan. Stall detection and influence of air injection on the performance of a single stage axial compressor. Master’s thesis, von Karman Institute for Fluid Dynamics, 2006. 33
- [37] C. Freeman. Effect of tip clearance flow on compressor stability and engine performances. In *VKI LS 1985-05 on Tip Clearance Effects in Axial Turbomachinery*, 1985. 12
- [38] H. Fujita and H. Takata. A study on configurations of casing treatment for axial flow compressor. *Bulletin of the JSME*, 27:1675–1681, 1984. 35
- [39] A. J. Gannon, G. V. Hobson, R. P. Shreeve, and I. J. Villescas. Experimental investigation during stall and surge in a transonic fan-stage and rotor only configuration. In *Proceedings of ASME Turbo Expo 2006*, 2006. 30, 121, 124, 159, 160
- [40] S. A. Gbadebo, N. A. Cumpsty, and T. P. Hynes. Three-dimensional separations in axial compressor. *Journal of Turbomachinery*, 127:331–339, 2005. 21
- [41] M. N. Goodhand and R. J. Miller. The impact of real geometries on three-dimensional separations in compressors. In *Proceedings of ASME Turbo Expo 2010*, number GT2010-22246, 2010. 24, 93
- [42] E. M. Greitzer. Surge and stall in axial flow compressors, part i and part ii. *Journal of Engineering for Power*, 98:190–217, 1976. 32, 201
- [43] E. M. Greitzer, J. P. Nikkanen, D. E. Haddad, R. S. Mazzawy, and H. D. Joslyn. A fundamental criterion for the application of rotor casing treatment. *Journal of Fluids Engineering*, 101:237–243, 1979. 25, 35, 36, 99, 106, 141, 177, 178, 201
- [44] C. Hah and J. Loellbach. Development of hub corner stall and its influence on the performance of axial compressor blade rows. *Journal of Turbomachinery*, 121:67–77, 1999. 20
- [45] E. J. Hall, A. J. Crook, and R. A. Delaney. Aerodynamic analysis of compressor casing treatment with 3-d navier-stokes solver. In *Joint Propulsion Conference, AIAA 94-2796*, 1994. 36
- [46] E. J. Hall, D. A. Topp, N. J. Heidelberg, and R. A. Delaney. Investigation of advanced counterrotation blade configuration concepts for high speed turboprop systems: Task vii - endwall treatment inlet flow distortion analysis final report. Technical report, Technical Report CR 195468, NASA, 1995. 36
- [47] A. Harten, B. Engquist, S. Osher, and S. Chakravarthy. Uniformly high order essentially non-oscillatory schemes. *Journal of Computational Physics*, 71:231–303, 1987. 180

- [48] M. D. Hathaway. Passive endwall treatments for enhancing stability. In *VKI LS 2006-06 on Advances in Axial Compressor Aerodynamics*, 2006. 34, 201
- [49] L. He. Computational study of rotating-stall inception in axial compressors. *Journal of Propulsion and Power*, 13:31–38, 1997. 158
- [50] M. Hembera, F. Danner, and H.-P. Kau. Development of circumferential grooves for axial compressors based on flow mechanisms. In *Joint Propulsion Conference, AIAA 2008-4988*, 2008. 39, 202
- [51] S. Hiernaux, C. Lebas, J.-F. Brouckaert, and I. Lepot. Next generation boosters. In *Proceedings of the XVIII ISABE Symposium*, 2007. 7
- [52] J. O. Hinze. *Turbulence*. McGraw-Hill Book Compagny, 1975. 82
- [53] T. Houghton and I. Day. Enhancing the stability of subsonic compressors using casing grooves. In *Proceedings of ASME Turbo Expo 2009*, number GT2009-59210, 2009. 40, 202
- [54] T. Houghton and I. Day. Stability enhancement by casing grooves: the importance of stall inception mechanism and solidity. In *Proceedings of ASME Turbo Expo 2010*, number GT2010-22284, 2010. 41, 100, 178
- [55] T. O. Houghton. *Axial Compressor Stability Enhancement*. PhD thesis, University of Cambridge, 2010. 102
- [56] D. A. Hoying, C. S. Tan, H. Vo, and E. M. Greitzer. Role of blade passage flow structure in axial compressor rotating stall inception. *Journal of Turbomachinery*, 121:735–742, 1999. 18, 29
- [57] M. C. Huppert. Preliminary investigation of flow fluctuations during surge and blade row stall in axial-flow compressors. Technical report, National Advisory Committee for Aeronautics, 1952. 158
- [58] M. Inoue and M. Kuroumaru. Structure of tip clearance flow in an isolated axial compressor rotor. *Journal of Turbomachinery*, 111:250–256, 1989. 13, 14, 16, 129, 136, 201
- [59] G. Jiang and C.-W. Shu. Efficient implementation of weighted eno schemes. *Journal of Computational Physics*, 126:202–228, 1996. 180
- [60] M. Kayabasi. Gas path temperature measurement in high temperature gas turbine environments. Master’s thesis, von Karman Institute for Fluid Dynamics, 2010. 66
- [61] S. A. Khalid, A. S. Khalsa, I. A. Waitz, C. S. Tan, E. M. Greitzer, N. A. Cumpsty, J. J. Adamczyk, and F. E. Marble. Endwall blockage in axial compressors. *Journal of Turbomachinery*, 121:499–509, 1999. 15, 16, 17, 39, 89, 201

- [62] S. J. Kline, J. G. Bardina, and R. C. Strawn. Correlation of the detachment of two dimensional turbulent boundary layers. *AIAA Journal*, 21:68–73, 1983. 82
- [63] C. C. Koch. Discussion of benser w. a.: Transonic compressor technology advancements. Technical report, NASA, SP-304, 1974. 130
- [64] C. C. Koch and L. H. Smith. Loss sources and magnitude in axial compressors. *Journal of Engineering for Power*, 98:411–424, 1976. 18
- [65] M. Laazareff. *elsA, Theoretical Manual*. ONERA - The French Aerospace Lab, version 2.1 edition, 2010. 54
- [66] B. Lakshminarayana and A. Pandya. Investigation of the tip clearance flow inside and at the exit of a compressor rotor passage - part 1: Mean velocity field. *Journal of Engineering for Power*, 105:1–12, 1983. 14, 15, 201
- [67] B. Lakshminarayana, M. Pouagare, and R. Davino. Three-dimensional flow field in the tip region of a compressor rotor passage-part1: Mean velocity and annulus wall boundary layer. *Journal of Engineering for Power*, 104:760–771, 1982. 14
- [68] B. E. Launder and B. I. Sharma. Application of the energy dissipation model of turbulence to the calculation of flow near a spinning disc. *Letter Heat Mass Transfer*, 1:131–138, 1974. 54
- [69] V.-M. Lei, Z. S. Spakovszky, and E. M. Greitzer. A criterion for axial compressor hub-corner stall. *Journal of Turbomachinery*, 130:031006 1–10, 2008. 21, 22, 24, 26, 201
- [70] Y. Levy and J. Pismenny. The number and speed of stall cells during rotating stall. In *Proceedings of ASME Turbo Expo 2004*, number GT2003-38221, 2003. 159, 165, 167
- [71] Y. Levy, J. Pismenny, A. Reissner, and W. Riess. Relationship between frequencies of pressure oscillations and rotor speed under rotating stall. *International Journal of Turbo and Jet Engines*, 19:233–247, 2002. 159
- [72] J. Lewalle. Application of continuous wavelets to data analysis, part 1: Wavelets without lemmas. In *VKI LS 1998-06 on Advanced Measurement Techniques*, 1998. 160, 161, 162
- [73] S. Lieblein. Experimental flow in two-dimensional cascades. Technical report, SP 36, NASA, 1975. 26, 99
- [74] F. Lin, J. Chen, and M. Li. Experimental investigation of unsteady rotor tip flows in a high speed compressor throttled to stall. In *Proceedings of ASME Turbo Expo 2002*, number GT2002-30360, 2002. 161, 162

- [75] F. Lin, J. Chen, and M. Li. Wavelet analysis of rotor-tip disturbances in an axial-flow compressor. *Journal of Propulsion and Power*, 20:319–334, 2004. 162
- [76] F. Lin, M. Li, and J. Chen. Long-to-short length scale transition: A stall inception phenomena in an axial compressor with inlet distortion. In *Proceedings of ASME Turbo Expo 2005*, number GT2005-68656, 2005. 30, 162
- [77] X.-D. Liu, S. Osher, and T. Chan. Weighted essentially non-oscillatory schemes. *Journal of Computational Physics*, 115:200–212, 1994. 180
- [78] X. Lu, W. Chu, J. Zhu, and Y. Wu. Mechanism of the interaction between casing treatment and tip leakage flow in a subsonic axial compressor. In *Proceedings of ASME Turbo Expo 2006*, number GT2006-90077, 2006. 17, 130
- [79] R. Mailach and K. Vogeler. Unsteady aerodynamic blade excitation at the stability limit and during rotating stall in an axial compressor. *Journal of Turbomachinery*, 129:503–511, 2007. 30, 159
- [80] C. Mathioudakis. *Rotating stall in axial flow compressors*. PhD thesis, Katholieke Universiteit Leuven - von Karman Institute for Fluid Dynamics, 1985. 31
- [81] F. K. Moore and E. M. Greitzer. A theory of post-stall transient in axial compression system: Part i- development of equations, and part ii - applications. *Journal of Engineering for Gas Turbines and Power*, 108:68–76, 1986. 28
- [82] R. D. Moore, G. Kovich, and R. J. Blade. Effect of casing treatment on overall and blade-element performance of a compressor rotor. Technical report, Technical Report TN D-6538, NASA, 1971. 34, 35
- [83] W. M. Osborn, G. W. Lewis, and L. J. Heidelberg. Effect of several porous casing treatment on stall limit and overall performances of an axial-flow compressor rotor. Technical report, Technical Report TN D-6537, NASA, 1971. 34, 35
- [84] R. C. Pampreen. Compressor surge and stall. Technical report, Concepts ETI, Inc, 1993. 28, 201
- [85] R. C. Pampreen. *Compressor Surge and Stall*. Concept ETI, Inc., 1993. 174, 177
- [86] G. Paniagua. *Investigation of the Steady and Unsteady Performance of a Transonic HP Turbine*. PhD thesis, Universit Libre de Bruxelles and von Karman Institute for Fluid Dynamics, 2002. 68

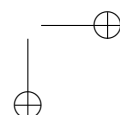
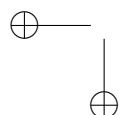
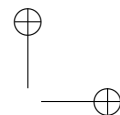
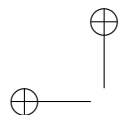
- [87] C. A. Poensgen and H. E. Gallus. Rotating stall in a single-stage axial flow compressor. *Journal of Turbomachinery*, 118:189–196, 1996. 158
- [88] D. C. Prince, D. D. Wisler, and D. E. Hilvers. Study of casing treatment stall margin improvement phenomena. Technical report, Technical Report CR134552, NASA, 1974. 34
- [89] D. A. Rains. Tip clearance flows in axial compressors and pumps. Master’s thesis, California Institute of Technology, 1954. 14
- [90] S. Reising, J. Font Brossa, and H.-P. Schiffer. Cfd analysis of hub-corner stall and secondary flow in a transonic compressor stage with non-axisymmetric end walls. In *Proceedings of the 8th European Conference on Turbomachinery*, 2009. 7
- [91] B. Roberts. Prediction of reynolds number effects on cascade performance. In *VKI Turbomachinery Laboratory Contributions - Lectures Given at Udine, Italy*, 1973. 91
- [92] M. Scervini and C. M. F. Rae. Thermocouple degradation mechanisms. In *VKI LS 2009-06 on Advanced High Temperature Instrumentation for Gas Turbine Application*, 2009. 65
- [93] S. G. Scheidler and F. Grauer. Experiment in active stall control of a twin-spool turbofan engine. In *Proceedings of ASME Turbo Expo 2002: Power for Land, Sea and Air*, 2002. 33
- [94] H. D. Schulz and H. D. Gallus. Experimental investigation of the three-dimensional flow in an annular compressor cascade. *Journal of Turbomachinery*, 110:467–478, 1988. 20
- [95] A. Shabbir and J. J. Adamczyk. Flow mechanism for stall margin improvement due to circumferential casing grooves on axial compressor. In *Proceedings of ASME Turbo Expo 2004: Power for Land, Sea and Air*, 2004. 36
- [96] A. Shabbir and J. J. Adamczyk. Flow mechanism for stall margin improvement due to circumferential casing grooves on axial compressors. *Journal of Turbomachinery*, 127:708–717, 2005. 37, 38, 201, 202
- [97] C. H. Sieverding. *Course Note 204 - Secondary flows in turbine blade passages*. von Karman Institute, 2004. 11
- [98] B. R. Smith. The k-l turbulence model and wall layer model for compressible flows. In *AIAA paper 90 - 1483*, 1990. 54
- [99] G. D. J. Smith and N. A. Cumpsty. Flow phenomena in compressor casing treatment. *Journal of Engineering for Gas Turbines and Power*, 106:532–541, 1984. 35

- [100] N. H. S. Smith. The aerodynamic design of the lp system. In *VKI LS on Aero-Engine Design: From State of the Art Turbofans Towards Innovative Architectures*, 2008. 1
- [101] Ames Research Staff. Equations, tables, and charts for compressible flow. Technical Report NACA 1135, Ames Aeronautical Laboratory, 1953. 66, 71, 84
- [102] J. A. Storer and N. A. Cumpsty. Tip leakage flow in axial compressors. *Journal of Turbomachinery*, 113:252–259, 1991. 16, 17, 129, 201
- [103] A. J. Strazisar and K. L. Suder. Compressor stall control through endwall recirculation. In *Proceedings of ASME Turbo Expo 2004: Power for Land, Sea and Air*, 2004. 33
- [104] K. L. Suder and M. D. Hathaway. Compressor stability enhancement using discrete tip injection. In *Proceedings of ASME Turbo Expo 2002: Power for Land, Sea and Air*, 2000. 33
- [105] H. Takata and Y. Tsukuda. Stall margin improvement by casing treatment - its mechanism effectiveness. *Journal of Engineering for Power*, 99:121–133, 1977. 35
- [106] C. S. Tan. Three-dimensional and tip clearance flows in compressors. In *VKI LS 2006-06 on Advances in Axial Compressor Aerodynamics*, 2006. 12, 13, 18, 91, 94, 105, 130, 154, 177, 201
- [107] N. Tantot and J. Julliard. From turbojet to innovative architectures: Open rotor and contra rotative fan engines. In *VKI LS on Aero-Engine Design: From State of the Art Turbofans Towards Innovative Architectures*, 2008. 1
- [108] **N. Van de Wyer**, B. Farkas, J. Desset, J.-F. Brouckaert, J.-F. Thomas, and S. Hiernaux. Experimental investigation of the steady and unsteady flow field in a single stage low pressure axial compressor with a circumferential groove casing treatment. In *Proceeding of ASME Turbo Expo 2010: Power for Land, Sea and Air*, 2010. 100, 103
- [109] C. Tropea. Processing of time-dependent data. In *VKI LS 2003-03 on Post-Processing of Experimental and Numerical Data*, 2003. 159, 161
- [110] H. Vo, C. S. Tan, and H. P. Grant. Criteria for spike initiated rotating stall. In *Proceedings of ASME Turbo Expo 2005: Power for Land, Sea and Air*, number GT 2005-68374, 2005. 18, 29, 30, 201
- [111] D. C. Wilcox. Formulation of the k-omega turbulence model revisited. *AIAA Journal*, 46:2823–2838, 2008. 54

- [112] G. L. Wilde. Improvement in or relating to gas turbines. british patent (701,576), filed june 28, 1950. 33
- [113] G. Wilfert. Geared fan. In *VKI LS on Aero-Engine Design: From State of the Art Turbofans Towards Innovative Architectures*, 2008. 4
- [114] I. Wilke and H.-P. Kau. A numerical investigation of the influence of casing treatment on the tip leakage flow in a hp front stage. In *Proceedings of ASME Turbo Expo 2002*, number GT2002-30642, 2002. 137
- [115] I. Wilke and H.-P. Kau. A numerical investigation of the flow mechanisms in a high pressure compressor front stage with axial slots. *Journal of Turbomachinery*, 126:339–349, 2004. 36
- [116] D. C. Wisler. Loss reduction in axial-flow compressors trough low-speed model testing. *Journal of Engineering for Gas Turbines and Power*, 107:354–363, 1985. 18
- [117] Z. Yang and T. H. Shih. New time scale based k-eps model for near-wall turbulence. *AIAA Journal*, 31:1191–1198, 1993. 54
- [118] X. Yu and B. Liu. A prediction model for corner separation/stall in axial compressors. In *Proceedings of ASME Turbo Expo 2010: Power for Land, Sea and Air*, number GT2010-22453, 2010. xi, 20, 22, 23, 24, 141, 149, 201

List of Tables

3.1. Stage characteristics of the VITAL test section	48
3.2. Stage characteristics of the VITAL test section	52
4.1. Tip- and hub displacement thickness at the inlet of the compressor stage for various operating points under cruise conditions	82
4.2. Tip- and hub momentum-loss thickness at the inlet of the compressor stage for various operating points under cruise conditions	83
4.3. Tip- and hub shape factor of the boundary layer at the inlet of the compressor stage for various operating points under cruise conditions	83
4.4. Summary of the experimental and numerical global performance of the compressor stage at two Reynolds numbers for the smooth casing and casing treatment configurations	108
6.1. Summary of the stall cells characteristics in the compressor stage during tests presenting various behavior, and for the smooth casing and casing treatment configurations	174



List of Figures

1.1. Innovative VITAL engine architectures and developments (generic and specific to the type of engine) [2].	3
1.2. Noise evolution in function of the By-Pass ratio.	5
1.3. Fuel consumption evolution in function of the By-Pass ratio. . . .	6
1.4. Phi/Psi diagram of booster stages.	7
2.1. Tip gap effects on the overall performance and stability of a high-speed six-stage compressor [106].	13
2.2. Tip leakage flow and resulting vortex from [58].	14
2.3. Velocity triangle in the tip clearance [66].	15
2.4. Casing wall pressure coefficient of a rotor in a multistage axial compressor for different tip gap size [102].	16
2.5. Influence of the clearance on the blockages in a low speed compressor [61].	17
2.6. Wall static pressure field of a transonic axial compressor under design and near stall conditions [9].	18
2.7. Deviation of the boundary layer flow in the blade passage.	19
2.8. Corner separation flow topology without (left) and with corner stall (right) [118].	20
2.9. Compressor stall indicator S in function of diffusion parameter D [69].	22
2.10. Compressor stall indicator S_W in function of diffusion parameter D_M [118].	23
2.11. Performance map.	25
2.12. Velocity triangle upstream the rotor for design and reduced mass flow conditions.	26
2.13. Spike stall inception process [84].	28
2.14. Modal disturbance in a stall inception process [84].	28
2.15. Stall Inception [24].	29
2.16. Criteria for spike inception [110].	30
2.17. Classical (left) and deep (right) surge phenomena [42].	32
2.18. Casing treatment configurations investigated in the 1970's [48]. . .	34
2.19. Compressor stall inception mode [43].	36
2.20. Pressure rise coefficient of a low speed axial compressor with and without casing treatment [96].	37
2.21. Radial velocity distribution in the casing plane [96].	38

2.22. Normalized axial velocity contours downstream the rotor trailing edge for two casing configuration and two flow conditions corresponding to conditions away from stall ($\phi = 0.3725$) and close to stall ($\phi = 0.35$) [96].	38
2.23. Flow mechanism in circumferential grooves [50].	39
2.24. Stall margin (SMI) and maximum efficiency (MEI) improvement in function of the groove casing treatment axial location [53].	40
3.1. Layout of the VKI-R4 closed loop high-speed compressor test rig .	44
3.2. Diagram of the available compressor power in function of the rotational speed of the R4 compressor test rig.	45
3.3. Layout of the VITAL test section	46
3.4. Photograph of the inlet of the VITAL test section	47
3.5. Circumferential distribution of the rotor tip clearance at leading and trailing edges at 100% and 105% nominal speed and after stall	48
3.6. The four axial positions of measurement station in the compressor stage	49
3.7. Probe radial traversing system installed on the test section	50
3.8. schematic view of the circumferential displacement mechanism and its encoder	51
3.9. Mesh topology around the rotor blade	53
3.10. Circumferential groove geometry	56
4.1. The head of the Prandtl probe	61
4.2. The head of the Kiel probe	62
4.3. Angular sensitivity of the total pressure probes	63
4.4. The head of the shielded thermocouple probe	65
4.5. Thermocouple measurement circuit with reference temperature area	65
4.6. Static calibration curves for the various thermocouples used in the VITAL project	67
4.7. Recovery factor as a function of the Mach number for the various thermocouples used in the VITAL project	68
4.8. Uncertainty on the recovery factor due to the uncertainty on the Mach number	70
4.9. The miniature three-hole probe used for the determination of the radial distributions of the velocity, flow angle and static pressure .	72
4.10. Three-holes probe schematic view	72
4.11. Calibration maps of the three-hole probe sensitivity coefficients as a function of Mach number and flow angle	74
4.12. The head of the hot-wire probe	75
4.13. Voltage given by a hot-wire as function of the flow velocity at constant density	78

4.14. Experimental radial distribution of the turbulence intensity at the stage inlet for the various throttling conditions under cruise pressure condition	79
4.15. Numerical radial distribution of the turbulence intensity at the stage inlet for the smooth casing and for the configuration with a cavity modeling the leakage around the probe stem for various leakage mass flows. Simulations with the code FINE by B. Farkas	80
4.16. Experimental radial distribution of the total pressure at the stage inlet non-dimensionalized by the mid-span pressure for the smooth casing configuration under cruise conditions at various operating points	81
4.17. Experimental and numerical (FINE) total pressure map at the outlet of the stage at design operating point under cruise conditions. .	87
4.18. Experimental (full line) and numerical (FINE - dashed line) circumferential distribution of the total-to-total pressure ratio P_{04}/P_{01} under cruise conditions at the design operating point at 10% (in red), 50% (in blue) and 90% (in black) of the blade height	89
4.19. Experimental total-to-total pressure ratio under cruise and take-off conditions at nominal speed, and numerical total-to-total pressure ratio at nominal speed under cruise conditions in function of the reduced mass flow	90
4.20. Experimental static-to-total pressure ratio at nominal speed under cruise and take-off conditions, and numerical static-to-total pressure ratio at nominal speed under cruise conditions	92
4.21. Experimental total-to-total efficiency at nominal speed under cruise and take-off conditions, and numerical total-to-total efficiency at nominal speed under cruise conditions	94
4.22. Groove around the hub of the stator blades	95
4.23. Radial distributions of the experimental and numerical total-to-total pressure ratios of the rotor under cruise conditions at design- (a) and near stall (b) operating points.	96
4.24. Radial distributions of the experimental and numerical total-to-total temperature ratios of the rotor under cruise conditions at design- (a) and near stall (b) operating points.	97
4.25. Radial distributions of the experimental and numerical total-to-total pressure ratios of the IGV under cruise conditions at near stall operating points.	98
4.26. Experimental diffusion factor of the rotor alone versus the non-dimensionalized pressure rise under cruise conditions over the entire operating range	99
4.27. Experimental and numerical total pressure ratio and relative efficiency versus the reduced mass flow under cruise conditions for the smooth casing and casing treatment configurations.	101

4.28. Experimental total-to-total pressure ratio at mid-span iso-speed versus the reduced mass flow for the smooth casing and the casing treatment configurations	102
4.29. Radial distribution of the experimental and numerical diffusion factor under cruise conditions for the smooth casing and casing treatment configurations at design (a) and near stall (b) operating points.	104
5.1. View of the XCS-062 sensor	110
5.2. Drawing of the XCS-062 sensor	110
5.3. Brass inserts for the Kulites	111
5.4. Brass inserts including the Kulites mounted inside the casing . . .	112
5.5. Schematic view of the cavity in front of the Kulite	112
5.6. Miniature fast response total pressure probe AP1-C25	113
5.7. Schematic view of casing static pressure measurement locations . .	114
5.8. Photograph of casing with static pressure measurement sensors . .	115
5.9. Circumferential arrangement of the sensors used for the stall cell detection	115
5.10. Schematic top view of the casing static pressure measurement locations for both casing configurations	116
5.11. Photograph from the bottom of the rotor casing with the unsteady static pressure measurement systems in the casing treatment configuration	117
5.12. Power spectral density of the unsteady casing wall pressure signals for cruise conditions under design point	118
5.13. Raw signal from the mid-chord fast pressure sensor K3 (top) and from the trigger (bottom) over one rotor revolution	119
5.14. Relative unsteady pressure at mid-chord over 5 blade passages . .	120
5.15. PLA of the mid-chord pressure signal compared to two random blade passages	121
5.16. 3D visualization of the phase-locked averaged casing wall static pressure under design conditions	122
5.17. Static pressure field, under design conditions for smooth casing configuration, based on an interpolation along the axial direction . . .	123
5.18. Static pressure field, under design (left) and near-stall (right) conditions for smooth casing configuration, based on an interpolation along the stagger angle direction	124
5.19. Experimental (dot) and averaged numerical (single line, FINE by B. Farkas) non-dimensionalized unsteady casing static pressure variation at design working point at various axial positions under cruise conditions	125
5.20. Experimental (dot) and numerical (single line, FINE by B. Farkas) non-dimensionalized unsteady casing static pressure variation at design working point at various axial positions	126

List of Figures 205

5.21. Experimental non-dimensionalized unsteady casing static pressure variation at design working point at various axial positions under cruise and take-off pressure conditions	127
5.22. Non-dimensional unsteady casing wall static pressure field from the interpolation of the experimental measurement (left) and from the CFD calculation using FINE by B. Farkas (right) in the smooth casing configuration under design working conditions	128
5.23. Non-dimensional unsteady casing wall static pressure field from the interpolation of the experimental measurement (left) and from the CFD calculation using FINE by B. Farkas (right) in the smooth casing configuration under near-stall working conditions	129
5.24. Non-dimensional experimental interpolated unsteady casing wall static pressure difference between the cruise and the takeoff conditions in the smooth casing configuration at near-stall working point	131
5.25. Non-dimensional experimental (left) and numerical (right, FINE by B. Farkas) casing static pressure field for smooth casing (top of the figures) and casing treatment configuration (bottom of the figures) under design working conditions (point <i>C</i>)	132
5.26. Non-dimensional experimental (left) and numerical (right, FINE by B. Farkas) casing static pressure field for smooth casing (top of the figures) and casing treatment configuration (bottom of the figures) under near-stall working conditions (point <i>D</i>)	133
5.27. Non-dimensional experimental and numerical (FINE by B. Farkas) static pressure at the top of the circumferential groove casing treatment under design working point (left) and near-stall working conditions (right)	134
5.28. Experimental non-dimensional casing wall static pressure fluctuations around the PLA (RMS) under choke working conditions for the smooth casing configuration	135
5.29. Experimental non-dimensional casing wall static pressure fluctuations (RMS) under near-stall working conditions for the smooth casing configuration (left) and the casing treatment configuration (right)	136
5.30. Experimental (top) and numerical (bottom, FINE by B. Farkas) non-dimensional pressure field in the planes of the casing and downstream the rotor, under design working conditions for a smooth casing configuration	138
5.31. Visualization by numerical streamlines of the tip leakage vortex and the flow separation on the suction side of the blade at the hub under design conditions for the smooth casing configuration, FINE by B. Farkas	139

5.32. Simultaneous visualization of the non-dimensional measured pressure field in the planes of the casing and downstream the rotor, under near-stall working conditions for a smooth casing (top) and casing treatment (bottom) configurations	140
5.33. Visualization by numerical streamlines of the tip leakage vortex and the flow separation on the suction side of the blade at the hub under near-stall conditions for the smooth casing configuration, FINE by B. Farkas	142
5.34. Simultaneous visualization of the non-dimensional numerical pressure field in the planes of the casing and downstream the rotor, under near-stall working conditions for a smooth casing (top) and casing treatment (bottom) configurations, FINE by B. Farkas . . .	143
5.35. Experimental non-dimensional pressure fluctuations field (RMS) in the planes of the casing and downstream the rotor under choke working conditions for a smooth casing configuration	144
5.36. Experimental non-dimensional pressure fluctuations field (RMS) in the casing and downstream rotor planes under near-stall working conditions for a smooth casing (top) and casing treatment (bottom) configurations	145
5.37. Difference of the experimental non-dimensional total pressure fluctuations (RMS) downstream the rotor between the two casing configurations under near stall working conditions	146
6.1. Axial and circumferential positioning of the fast pressure sensors used for stall cell detection	152
6.2. Axial arrangement of hot-wires used for stall cell detection and characterization	152
6.3. Fast pressure sensor K7 voltage raw signal during stall inception process for smooth casing configuration.	153
6.4. Voltage raw signal from fast pressure sensor located at different annular position during stall inception process for smooth casing configuration.	154
6.5. Low-pass filtered voltage signals from fast pressure sensors located at different annular positions during stall inception process for the smooth casing configuration.	155
6.6. Low-pass filtered voltage signal from fast pressure sensors installed at different annular position during stall inception process for the casing treatment configuration.	156
6.7. Voltage signal around the stall cell inception point from hot-wires.	157
6.8. Raw voltage signal from fast pressure sensor <i>K7</i> for three different experiments during rotating stall in smooth casing configuration.	158
6.9. Zoom on the time evolution of the raw voltage signal from fast pressure sensor <i>K7</i> in smooth casing configuration.	159
6.10. Shape of Mexican hat wavelet.	161

6.11. Shape of complex Morlet wavelet.	162
6.12. Intermittent signal (above) and its wavelet transformation (below).	163
6.13. Spectral density of the fast pressure signal for three experiments.	164
6.14. Wavelet transformation of the unsteady pressure signal for three experiments.	165
6.15. Idealized pressure signals of stall cells from three sensors at different circumferential locations and line of constant phase of pressure oscillation for a single stall cell on the left and a two stall cells case on the right.	166
6.16. Part <i>II</i> of the fast sensors voltage signals in experiment 1.	167
6.17. Transition between parts <i>II</i> and <i>III</i> of the fast sensors voltage signals in experiment 1.	168
6.18. Transition between parts <i>II</i> and <i>III</i> of the fast sensors voltage signals in experiment 3.	169
6.19. Simultaneous signal from static pressure sensor and hot wire under stall condition.	170
6.20. Determination of the stall cell extension based on graphical observation in experiment 1.	171
6.21. Determination of the stall cell extension based on graphical observation in experiment 2.	172
6.22. Determination of the stall cell extension based on graphical observation in experiment 3.	172
 B.1. Experimental (left) and numerical (right) casing static pressure field for smooth casing (top of the figures) and casing treatment configuration (bottom of the figures) under choke working conditions (point <i>C</i>)	185
B.2. Simultaneous visualization of the experimental pressure field in the planes of the casing and downstream the rotor, under choke working conditions for a smooth casing (top) and casing treatment (bottom) configurations	186
B.3. Simultaneous visualization of the numerical pressure field in the planes of the casing and downstream the rotor, under choke working conditions for a smooth casing (top) and casing treatment (bottom) configurations	187