UNIVERSIDAD POLITÉCNICA DE MADRID

ESCUELA TÉCNICA SUPERIOR DE INGENIERÍA

AERONÁUTICA Y DEL ESPACIO

Fourier Methods for Modelling of
Instabilities and Unsteady Flows in
Turbomachinery

TESIS DOCTORAL

David Romera Hijano

Máster en Ingeniería Aeronáutica

Madrid, 2020

Departamento de Mecánica de Fluidos y Propulsión

Aeroespacial

Escuela Técnica Superior de Ingeniería Aeronáutica y

del Espacio

Fourier Methods for Modelling of
Instabilities and Unsteady Flows in
Turbomachinery

Autor

David Romera Hijano

Máster en Ingeniería Aeronáutica

Director

Roque Corral García

Doctor Ingeniero Aeronáutico

Madrid, 2020

Escuela Técnica Superior de Ingeniería Aeronáutica y del Espacio

UNIVERSIDAD POLITÉCNICA DE MADRID

Tribunal nombrado por el Magnífico y Excelentísimo Señor Rector de la Universidad Politécnica de Madrid. El día   de   de 2020.

Presidente: D. Javier Jiménez Sendín

Secretario: D. Carlos Martel Escobar

Vocal: D. Mehdi Vahdati

Vocal: D. Christian Frey

Vocal: D. Almudena Vega Coso

Suplente: D. Manuel Antonio Burgos Olmos

Suplente: D. Raquel Gómez Miguel

Realizado el acto de defensa y lectura de la tesis el día   de   de 2020 en la E.T.S.I. Aeronáutica y del Espacio.

El Presidente

Los Vocales

El Secretario

Acknowledgements

At the end of my Ph.D research, I would like to express my sincerely gratitude to all those people who have been helping me through these years. It may not possible to mention every individual's name, but I do like to sincerely thank everyone who has shared time with me, inside and outside of work.

First, I would like to express my greatest gratitude to my Ph.D supervisor Prof. Roque Corral. Thanks for providing me the opportunity to work and do my research in ITP Aero. He has been providing many guidance and advices in all time of my study. Much of my understanding of the subject, the project and the code is owed to him. His immense knowledge and enthusiasm in engineering work and research are worth to learn for my entire life.

I would also like to thank my research colleagues Michele Greco, Miguel Montiel, Ricardo Blázquez, Marc Bolinches and Yago Blando for making the last four years merry ones. Those coffee breaks and debates at lunch time deserve to be acknowledged.

I also gratefully acknowledge ITP Aero for providing financial support toward this research. My thanks are dedicated to the directors and work colleagues in the Advanced Engineering Direction, for their supports to work as a part of this talent team. I would specially thank Jaime Fernández Castañeda as Head of Research & Technology for all the support along these years. Additionally, particular thanks must go to Javier Crespo, Jesús Contreras, Jesús Pueblas and Fernando Gisbert for giving me many advices in performing numerical simulations, Juan Mauricio Chia and Jaime Quintanal for their help in grid generation systems, Carlos Perez for his knowledge in rotating cavities, Eduardo Garcia and Álvaro González for the management in cluster systems. And many more, Juan Manuel Gallardo, Oscar Bermejo, Jose María Chaquet,

Olga García, etc. I received so many helps from all of you, in work, and in life. These years of working experience in ITP Aero will certainly be one of the most valuable and unforgettable time.

Finally, I am extremely grateful to my parents and my brother for their support, encouragement and everything else. Words can not express how grateful to them. Their loves and understanding sustained me thus far.

Last but not the least, I would like to thank all the jury members: Prof. Javier Jiménez, Prof. Carlos Martel, Prof. Mehdi Vahdati, Prof. Christian Frey and Dr. Almudena Vega for their presence, evaluation and providing many valuable advices to my work.

This research has been carried-out under the RETOS project RTC1814120123 named EN-VIDIA the grant from the Spanish Ministerio de Economía y Competitividad DPI2017-84700-R named AEROMOT.

Abstract

This thesis presents the formulation and validation of a novel time-marching Passage-Spectral Method (PSM) for turbomachinery applications. The proposed method provides an efficient means of approximating flows with spatial Fourier methods based on blocks, trying to facilitate the efficient analysis for design and optimization of unsteady and non-axisymmetric flows, reducing the computational cost of these expensive simulations and simplify the study of travelling disturbances without making any hypothesis about its temporal periodicity.

Improving engine performance, reducing the cost of new products and complying with environmental regulations are areas of intense interest for the turbomachinery industry. Computational Fluid Dynamics (CFD) solutions in combination with theoretical approaches can improve engine design by offering a powerful way to develop a physical understanding of the flow phenomena in these systems. However, they have encountered a number of challenges for unsteady and non-axisymmetric problems in turbomachinery, since the complexity of some characteristics such as geometries, flow conditions and rotating instabilities. The prediction of three-dimensional instabilities is a subject of both fundamental and practical importance in fluid mechanics. The identification and characterization of mechanisms related to this process should improve prediction methods and lead to new, more efficient control strategies both considerable importance in practical flows. Numerical simulations of non-axisymmetric and unsteady flows are normally conducted using full-annulus geometries and Unsteady Reynolds-averaged Navier-Stokes (URANS) solvers.

In general, non-linear flow disturbances in time and/or space can be identified and represented by a discrete Fourier series as long as they are periodic. This thesis focuses on the challenges

in the numerical simulation of these complex flows, proposing an efficient block-wise spatial Fourier method to take advantages of the spatial periodicity of some rotating disturbances. The new method is intended for problems with a short characteristic length perturbation with a superimposed long-wavelength wave which will be Fourier transformed. The approach discretizes the domain using a finite number of blocks or passages, where the flow variables at the supposedly periodic boundaries are continuously updated using the spatial Fourier coefficients of a uniformly spaced set of blocks. First, the formulation and methodology is described for one-dimensional wave equation. A detail explanation on how to implement this novel strategy is carried out. The block-wise spectral reconstruction makes possible to accurately solve a wide range of spatial harmonics and the interaction between them, with the ultimate goal to reproduce long length-scale unsteady flow features of interest with a drastic reduction in number of reduced-passages to be required, numerically solving long length-scale unsteady inside each block without any Fourier transform. A third family of scattered harmonics is found to be solves with the same set of sampling blocks, making the proposed method very well suited for turbomachinery problems.

The method is then extended to the modelling of three-dimensional non-axisymmetric geometries and implement it in a finite-volume compressible flow solver. The thesis aims to provide evidence comparing results from the proposed PSM and full-annulus simulations for different practical applications, setting the boundaries and limitations of the novel strategy. This kind of methodology is applicable to the simulation of compressor instabilities (rotating stall) and the long-wavelength instabilities which can be seen in the disk drum and under-platform cavities of turbines and compressors where the associated frequencies are self-excited, which should be part of the solution.

The NASA rotor 67, which is a low-aspect-ratio fan, has been chosen as the verification vehicle of the proposed method due to the existence of experimental data and numerical simulations in the public domain. The NASA rotor 67 under the effect of clean and distorted inflow conditions has been used first as validation case to demonstrate the effectiveness and viability of the method into non-axisymmetric case comparing against full-annulus solutions. The goal is to use

a few number of reduced-passages to capture the large-scale non-axisymmetric perturbations of the flow during the stall process under distorted operating conditions, filtering the highest harmonics which might not be of interest. The comparison between the Passage-Spectral Method and the full-annulus solution shows that sound solutions can be obtained with a low number of harmonics. The method is shown to accurately reproduce the full-annulus solution with a few spatial harmonics, capturing the characteristic features of the complex flow induced by the tip leakage vortex breakdown. The dependence of the aerodynamic stability of fan blades with the nodal diameter and amplitude of inlet and outlet perturbations is studied also using the potential of the proposed method. The method is further modified to deal efficiently with a single spatial harmonic and then applied to assess the effect of inlet and outlet distortions. Despite the complexity of the inlet distortion screen or the number of struts or pylons at the outlet, any inlet or outlet distortion pattern can be Fourier decomposed. The explicit spatial Fourier approximation is exploited to characterise the relevance of each nodal diameter of outlet perturbations in the fan stall process, and its non-linear stability is studied in a harmonic by harmonic basis. The stability of the spatial harmonics content can be assessed separately using the proposed PSM filtering the contribution of the rest of the circumferential modes of the imposed perturbation, retaining just three reduced-passages.

The last part of the thesis provides another application of the proposed numerical method in real turbomachinery configurations. The flow-field behaviour of different engine representative rotating cavities has also been investigated and presented, capturing rotating instabilities in different geometries of rotating cavities, which can be met in aero-engines due to system rotation. The rotating flows presents a non-axisymmetric behaviour under some operating conditions, affecting the flow patterns drastically compared to stationary cases. Basic flows produced by rotating disks, and their stability, will be reviewed as well, validating the proposed method in these complex flows with the corresponding savings in computational resources.

The results demonstrate that the computational time has been substantially reduced compared to full-annulus simulations, with good agreement in the results, which is attractive for industrial applications.

Keywords: computational fluid dynamics, unsteady flows, Fourier methods, fan-stage stability, rotating flows, heat transfer.

Resumen

Esta tesis presenta la formulación y validación de un novedoso Passage-Spectral Method (PSM) en el tiempo para aplicaciones de turbomaquinaria. El método propuesto proporciona un medio eficiente de aproximación de flujos con métodos espaciales de Fourier basados en bloques, tratando de facilitar un análisis eficiente para el diseño y optimización de flujos inestables y no simétricos, reduciendo el coste computacional de estas simulaciones y simplificando el estudio de perturbaciones no-estacionarias sin ninguna hipótesis sobre su periodicidad temporal.

Mejorar el rendimiento del motor, reducir el coste de nuevos productos y cumplir con las regulaciones ambientales son áreas de gran interés para la industria de turbomaquinaria. Las soluciones de Computational Fluid Dynamics (CFD) en combinación con enfoques teóricos pueden mejorar el diseño del motor al ofrecer un método potente para comprender la complejidad de estos fenómenos físicos de flujo en estos sistemas. Sin embargo, se han encontrado con una serie de desafíos para problemas inestables y no simétricos en turbomáquinas, debido a la complejidad de algunas características tales como geométricas, condiciones de flujo e inestabilidades de rotación. La predicción de inestabilidades tridimensionales es un área de amplia importancia práctica y teórica en mecánica de fluidos. La identificación y caracterización de los mecanismos relacionados con este proceso debería mejorar los métodos de predicción y conducir a nuevas estrategias de control más eficientes, ambas de considerable importancia en flujos reales. Las simulaciones numéricas de flujos inestables y no simétricos se realizan normalmente utilizando geometrías de corona completa y códigos Unsteady Reynolds-averaged Navier-Stokes (URANS).

En general, las perturbaciones de flujo no lineales en el tiempo y/o en el espacio pueden identi-

ficarse y representarse mediante una serie discreta de Fourier siempre que sean periódicas. Esta tesis se centra en los desafíos que supone la simulación numérica de estos flujos complejos, proponiendo un método eficiente espacial de Fourier por bloques para aprovechar la periodicidad espacial de algunas perturbaciones no estacionarias. El nuevo método está destinado a problemas con una perturbación de longitud característica de onda corta con una de longitud superpuesta de onda larga que será transformada en series de Fourier. Este nuevo enfoque discretiza el dominio usando un número finito de bloques o pasajes, donde las variables de flujo en los límites supuestamente periódicos se actualizan continuamente usando los coeficientes espaciales de Fourier a través de un conjunto de bloques espaciados uniformemente. Primero, se describe la formulación y metodología para la ecuación de onda unidimensional. Se realiza una explicación detallada de cómo implementar esta nueva estrategia. La reconstrucción espectral por bloques hace posible resolver con precisión una amplia gama de armónicos espaciales y la interacción entre ellos, con el objetivo final de reproducir características de interés de flujo inestable en una gran escala de longitudes de onda con una reducción drástica en el número de pasajes necesarios, resolviendo numéricamente la inestabilidad de onda larga dentro de cada bloque sin necesidad de usar ninguna transformación por Fourier. El método es capaz a su vez de resolver una tercera familia de armónicos con el mismo conjunto de bloques, lo que hace que el método propuesto sea muy adecuado para problemas de turbomaquinaria.

Posteriormente, el método se amplía al modelado de geometrías tridimensionales no simétricas y se implementa en un código de flujo compresible basado en volúmenes finitos. La tesis tiene como objetivo proporcionar evidencias sobre la comparación de los resultados del PSM propuesto y las simulaciones de corona completa para diferentes aplicaciones prácticas, estableciendo las limitaciones del nuevo enfoque. Este tipo de metodología es aplicable a la simulación de inestabilidades en compresores (entrada en pérdida) y las inestabilidades de longitud de onda larga que se pueden observar en la parte interna del disco y las cavidades debajo de la plataforma de turbinas y compresores donde las frecuencias asociadas se autoexcitan, lo que debería ser parte de la solución.

El NASA rotor 67, que es un fan de baja relación de aspecto, ha sido elegido como vehículo de

verificación del método propuesto debido a la existencia de datos experimentales y simulaciones numéricas de dominio público. El NASA rotor 67 bajo el efecto de condiciones de flujo de entrada limpias y distorsionadas se ha utilizado primero como caso de validación para demostrar la efectividad y viabilidad del método en un caso no simétrico en comparación con soluciones de corona completa. El objetivo es utilizar un número reducido de pasajes para capturar las perturbaciones no simétricas del flujo a gran escala durante el proceso de bloqueo en condiciones de funcionamiento distorsionadas, filtrando los armónicos más altos que podrían no ser de interés. La comparación entre el Passage-Spectral Method y la solución de corona completa muestra que se pueden obtener soluciones fiables con un número bajo de armónicos. Se muestra que el método es capaz de reproducir con precisión la solución de corona completa con algunos armónicos espaciales, capturando los rasgos característicos del flujo complejo inducido por la ruptura del vórtice de fuga de la punta. También se estudia la dependencia de la estabilidad aerodinámica de las palas del fan con el diámetro nodal y la amplitud de las perturbaciones de entrada y salida utilizando el potencial del método propuesto. El método se modifica a su vez para tratar de manera eficiente utilizando sólo un armónico espacial y luego se aplica para evaluar el efecto de las distorsiones de entrada y salida. A pesar de la complejidad de la distribución de la distorsión a la entrada o del número de estructuras o pilones en la salida, cualquier patrón de distorsión de entrada o salida puede descomponerse en series de Fourier. La aproximación espacial explícita de Fourier se explota para caracterizar la relevancia de cada diámetro nodal de las perturbaciones de salida en el proceso de parada del fan, y su estabilidad no lineal se estudia en una base armónico a armónico. La estabilidad del contenido de armónicos espaciales se puede evaluar por separado utilizando el PSM propuesto que filtra la contribución del resto de los modos circunferenciales de la perturbación impuesta, reteniendo solo tres pasajes.

La última parte de la tesis proporciona otra aplicación del método numérico propuesto en configuraciones reales de turbomaquinaria. También se ha investigado y presentado el comportamiento del campo de flujo en diferentes cavidades representativas, capturando inestabilidades no estacionarias en diferentes geometrías que se pueden encontrar en motores aeronáuticos debido a la rotación del sistema. Los flujos rotativos presentan un comportamiento no simétrico

en algunas condiciones de operación, afectando los patrones de flujo drásticamente en comparación con los casos estacionarios. También se revisarán los flujos básicos producidos por los discos rotativos, y su estabilidad, validando el método propuesto en estos flujos complejos con el correspondiente ahorro en recursos computacionales.

Los resultados demuestran que el tiempo de cálculo se ha reducido sustancialmente en comparación con las simulaciones de corona completa, con una buena comparación en los resultados, lo que resulta atractivo para aplicaciones industriales.

Palabras clave: dinámica de fluidos computacional, flujos no-estacionarios, métodos de Fourier, estabilidad en fanes, fluidos rotatorios, transferencia de calor.

Contents

List of Figures	xiv
List of Tables	xv
Nomenclature	xxi
1. Introduction	1
1.1. Background and Motivation	2
1.1.1. Unsteady Flows in Fan-stages	3
1.1.2. Rotating Instabilities in Cavities	5
1.2. Numerical Methods Challenges in Turbomachinery	7
1.2.1. Linear Frequency-domain Methods	9
1.2.2. Non-linear Frequency-domain Methods	10
1.2.3. Non-linear Time-domain Methods	12
1.2.4. Non-linear Methods for Non-axisymmetric Flows	15
1.3. Objectives and Outline	21
2. Fundamentals of Passage-Spectral Method	23
2.1. One-dimensional Linear Wave Equation Model	24
2.1.1. Motivation and Qualitative Description of the Method	26
2.1.2. Method Harmonic Content	27
2.1.3. Numerical Method Outline	29
2.1.4. Spectral View of Block Reconstruction	33
2.1.5. Numerical Results for the LWE	36

2.1.6.	Solution of the Wave Equation with varying propagation speed	45
2.2.	Generalization to 3D Non-axisymmetric Geometries	50
2.2.1.	Baseline Flow Solver	51
2.2.2.	3D Block-wise Spectral Method	54
2.2.3.	Practical Implementation	57
2.3.	Conclusions	59
3.	Unsteady Flows in a Fan-Stage	61
3.1.	Forcing Due to Inlet Distortion	62
3.2.	Rotating Stall Inception	71
3.2.1.	Clean Inflow	74
3.2.2.	Distorted Inflow	79
3.3.	Conclusions	91
4.	Non-linear Stability Analysis of a Generic Fan	93
4.1.	Modified Block Spectral Method	94
4.2.	Inlet Total Pressure Perturbation Stability Analysis	96
4.2.1.	Fan-Stage Stability	98
4.2.2.	Results	104
4.3.	Outlet Static Pressure Perturbation Stability Analysis	113
4.3.1.	Fan-Stage Stability	114
4.3.2.	Results	120
4.4.	Proposed Methodology	124
4.5.	Conclusions	125
5.	Unsteady Flows in Rotating Cavities	127
5.1.	Results for Rotating Cavities	128
5.1.1.	Rotor-Stator Cavity	130
5.1.2.	Compressor Cavity	139
5.1.3.	Rim Seal Cavity	145

5.2. Conclusions	154
6. Conclusions	157
Bibliography	161
A. Harmonic Content of Non-uniform Mean Flow Velocity Wave Equation	179
B. Modelling of Unsteady Fluid Flow and Heat Transfer in a Rotor-Stator Cavity	183
B.1. One-Dimensional Analytical Model	184
B.1.1. Governing Equations	184
B.1.2. Results	191
B.1.3. Evaluation of Unsteady Effects	208
B.2. Comparison with two-dimensional axisymmetric simulations	210
B.2.1. Steady solution	210
B.2.2. Unsteady solutions	212
B.3. Summary	217

List of Figures

1.1.	Perspective view of the Rolls-Royce UltraFan aero-engine.	3
1.2.	Instantaneous entropy contour after stall inception.	4
1.3.	Schematic diagram of non-axisymmetric flow structures in a rotating cavity core.	6
1.4.	Single passage domain with periodic boundaries.	12
1.5.	Computational mesh cell and its circumferential neighbours points.	16
1.6.	Computational domain for circumferential Fourier model [1].	18
2.1.	Comparison of the PSM and the analytical solution ($k_L = 1$ and $k_c = N_B = 9$).	26
2.2.	Schematic view of PSM harmonic content.	28
2.3.	Sketch of the node and block index notation used in the one-dimensional numerical scheme.	30
2.4.	Schematic of the block-wise spectral reconstruction of the 1 st harmonic ($k_L = 1$) in a domain consisting of six blocks, $N_B = 6$	31
2.5.	Sketch of the spectrum content of the method for different values of the number of spatial harmonics, M , retained in the Fourier reconstruction.	35
2.6.	Comparison of the PSM and full-domain solutions with an initial solution with $k_L = 2$. ($M = 2$, five computational blocks in white, and $N_B = 20$).	36
2.7.	Comparison of the PSM ($M = 2$) and the full-domain solutions with and initial solution dcontaining two harmonics ($k_L = 2$ and $k_S = 15$). The total number of blocks is $N_B = 15$	37

2.8. Comparison of PSM ($M = 1$, three computational blocks) and full-domain solutions (a) and harmonic Fourier content (b) with an initial solution described in Eqn. 2.19, where $k_L = 1$, $N_B = 9$, $\alpha = 0.2$ and $a = 10$	39
2.9. Comparison of PSM ($M = 1$, three computational blocks) and full-domain solutions (a), PSM-Reconstructed solution (b) and harmonic Fourier content (c) with the initial solution described in Eqn. 2.19, where $k_L = 1$, $N_B = 10$, $\alpha = 0.2$ and $a = 10$	41
2.10. Comparison of PSM ($M = 2$, five computational blocks) and full-domain solutions with an initial solution described in Eqn. 2.18 where $k_L = 2$, $k_S = 13$, $N_B = 15$ and $\alpha = 0.4$	43
2.11. Comparison of the PSM ($M = 2$) and the full-domain solutions with the initial solution described in Eqn. 2.18 where $k_L = 2$, $k_S = 5$, $N_B = k_c = 15$ and $\alpha = 0.4$	44
2.12. Analytical solution (a) in Eqn. 2.23 and harmonic Fourier content (b) with $k_L = 1$, $k_c = 6$ and $\beta = 0.5$	46
2.13. Comparison of the PSM ($M = 1$, three computational blocks), the full-domain and the analytical solutions (sub-figures (a) and (b)) and their harmonic contents (sub-figures (c) and (d)) for the case $\varepsilon = 0.8$ and $k_L = 1$, and two different wavenumbers of the propagation speed, $k_c = 4$ and $k_c = 6$. The wavelengths of the propagation speed and the block size are identical, $N_B = k_c$	49
2.14. Hybrid cell grid and associated dual mesh.	51
2.15. Schematic of a 1ND sinusoidal distribution in whole annulus (left) and its approximation (right) with three reduced-passages samples (dark grey) uniformly spaced and not in the physical position of the original full-annulus.	55
2.16. Updating periodic ghost cells.	56
3.1. NASA rotor 67 semi-unstructured grid (a) and computational domain (b). . . .	63
3.2. NASA rotor 67 meridional semi-unstructured mesh and full-annulus model. . .	64
3.3. NASA rotor 67 full computational domain with 3 reduced-passages ($M = 1$) approximation.	66

3.4. NASA rotor 67 speedline from PSM simulations.	66
3.5. Total pressure distribution at the fan inlet in the stationary frame of reference. .	67
3.6. PSM solution with $M = 1$ of the total pressure contours at 0.5 axial chords downstream of the rotor trailing-edge (see Fig. 3.9) at two different time instants for the forced response distortion case.	68
3.7. Axial location of screenshots shown in Fig. 3.6.	68
3.8. Comparison of blade loading at 80% span between the PSM with a single harmonic (dashed lines) and full-annulus solutions ($\circ$) at three different time instants over a cycle of the inlet disturbance. The clean inlet distribution ($\diamond$) is added as reference.	69
3.9. NASA rotor 67 semi-unstructured grid and numerical probe location.	70
3.10. Spatial Fourier decomposition of the absolute total pressure near the outer casing 0.5 axial chords downstream of fan trailing-edge (see Fig. 3.9) in the rotor frame of reference non-dimensionalised with the inlet total pressure amplitude, ΔP_0	70
3.11. Schematic rotating stall process.	72
3.12. Long NASA rotor 67 full computational domain.	73
3.13. Entropy distribution of tip region (99% span) after 16 rotor revolutions.	74
3.14. Snapshots of the relative Mach number at the mid-chord of the fan rotor during stall inception at "Near Stall" operating conditions, see point B ($\circ$) in Fig. 3.4.	76
3.15. Comparison of the relative isentropic Mach number at 80% span between PSM ($M = 1$) and full-annulus solutions subjected to rotating stall after 16 rotor revolutions. The "Near Stall" steady distribution ($\diamond$) is added as reference.	77
3.16. Relative isentropic Mach number contours of the three reduced-passages used in PSM ($M = 1$) solution subjected to rotating stall inception after 16 rotor revolutions.	78
3.17. Propagation of rotor upstream axial velocity disturbance at 0.5 axial chords of the fan trailing-edge and 99% span from hub for the PSM solution with $M = 1$	79

3.18. Top: Comparison of the full-annulus and the PSM transient spatial Fourier decomposition of the total pressure in a probe near the outer casing at 0.5 axial chords downstream of fan trailing-edge (see Fig. 3.9) non-dimensionalised with the inlet total pressure amplitude, ΔP_0 . Bottom: non-dimensional relative angular phase velocity of the spatial harmonics, Ω_m/Ω_{rotor} , at the final saturated state.	80
3.19. Axial velocity contours at 0.5 axial chords downstream of the fan trailing-edge (see Fig. 3.9) after 20 rotor revolutions for the low total pressure amplitude distortion case at the final saturated state.	81
3.20. Mass flow history at the outlet as a function of the time resolution.	84
3.21. Time required to achieve a stall mass flow of reference ($\dot{m}_{ref} \simeq 31.1 \text{ kg/s}$) as a function of the time resolution for the PSM with $M = 3$	84
3.22. Full-annulus Fourier decomposition of the relative axial velocity near the outer casing at 0.5 axial chords downstream of the fan trailing-edge (see Fig. 3.9) non-dimensionalised with the inlet reference axial velocity for the high total pressure amplitude distortion case. Solid lines: full-annulus solution. Dashed lines: PSM with $M = 3$	86
3.23. Axial velocity contours at 0.5 axial chords downstream of the fan trailing-edge (see Fig. 3.9) at two different time instants for the high total pressure amplitude distortion case at the final saturated state.	87
3.24. Instantaneous static pressure contours of the PSM ($M = 3$) solution at 60% span from hub.	90
4.1. Sketch of the location of three-reduced passages ($N_J = 3$) to represent an inlet perturbation with $M = 3$	94
4.2. Effect of inlet distortion amplitude at 100% of the design speed for $ND = 1$ computed with the PSM.	100

4.3.	Stability conceptual map due to different amplitudes of inlet distortion distribution at 100% of the design speed and nodal diameters, ND , using the proposed PSM just retaining the harmonic $M = ND$	102
4.4.	Stability conceptual map with clean inflow using PSM with a single harmonic M at 100% of the design speed.	103
4.5.	Full-annulus comparison between stable and unstable conditions with a first nodal diameter ($ND = 1$) perturbation with amplitudes of $\Delta P_0 = 1000 Pa$ and $\Delta P_0 = 3500 Pa$, marked as [a] and [b] in Fig. 4.3 (a) respectively, for the 1 st harmonic component ($m = 1$) of the total pressure signal at 0.5 axial chords downstream of fan (see Fig. 3.9). $P_0 = 101325 Pa$	105
4.6.	Fourier approximation (left) and total pressure inlet distortion profile (right) for the 120° and 60° screens with 15% and 25%, respectively of inlet total pressure deficit, and the equivalent sinusoidal perturbation. ($P_{0,ref} = 101325 Pa$).	106
4.7.	Comparison for the three different inlet distortion screens at 100% of the design speed. (a) and (d): sinusoidal screen, (b) and (e): 120°-screen, (c) and (f): 60°-screen. Top: Snapshot of the relative total pressure contours 0.5 axial chords downstream of the fan after 8 rotor revolutions non-dimensionalised with the inlet total pressure amplitude of the 1 st spatial harmonic, ΔP_0 . Bottom: non-dimensional relative angular phase velocity of the spatial harmonics, Ω_m/Ω_{rotor} , for the three inlet distortion screens of a total pressure probe near the outer casing 0.5 axial chords downstream of the fan (see Fig. 3.9) non-dimensionalised with ΔP_0	108
4.8.	Comparison between distorted and clean inflow after 8 rotor revolutions, once the solution reaches a quasi-steady state stall conditions with a first nodal diameter ($ND = 1$) perturbation with amplitude $\Delta P_0 = 3500 Pa$, marked as [a] in Fig. 4.3(a), for the 1 st harmonic component ($m = 1$) of the total pressure signal at 0.5 axial chords downstream of fan (see Fig. 3.9).	112

4.9. Stability conceptual map due to different amplitudes of static pressure patterns at the outlet and nodal diameters using the PSM just retaining the harmonic $M = ND$ at 100% of the design speed.	118
4.10. Fourier approximation (left) and static pressure outlet profile (right) for two 7°-sector static pressure perturbation with 30% of outlet static pressure deficit and the equivalent sinusoidal $ND = 2$ potential effect. ($p_{s,ref} = 120000 Pa$).	120
4.11. Full-annulus comparison between stable and unstable conditions with two 7°-sector static pressure perturbation at the outlet for two total pressure ratio of $PR = 1.64$ and $PR = 1.65$ marked as [a] and [b] in Fig. 4.9 (a), respectively. Mass flow comparison (left) and pressure signal (right) at 0.5 axial chords downstream of fan near the casing (see Fig. 3.9) for the 2 nd harmonic component ($m = 2$), non-dimensionalised with the distortion amplitude at the outlet, $\Delta p_s = 2500 Pa$	122
5.1. Sketch of disk cavities in a gas turbine [2].	128
5.2. A generic high-pressure gas turbine stage with rim seal ingestion flow (a) and rotating instabilities in the azimuthal direction at the inner part of the rotor-stator cavity core (b).	130
5.3. Structured mesh and main geometrical parameters of a generic rotor-stator cavity, where $R_0 = 100 mm$ and $R_i = 20 mm$	132
5.4. Reduced-passage sample mesh of the rotor-stator cavity with a sector size of 6°.	133
5.5. Sketch of the flow pattern in the inner part of a rotor-stator cavity.	134
5.6. Transition diagram by Schouveiler et al. [3] for large clearance and laminar flow. Curves labeled 'CR' indicate threshold for circular rolls, where the solution becomes non-axisymmetric.	136
5.7. Dimensionless radial velocity contours on the mid-cavity axial plane for a stable and unstable case, corresponding to points [a] and [b] in Fig. 5.6 respectively, after 20 rotor revolutions.	136

5.8. Comparison of dimensionless radial velocity contours on the mid-cavity axial plane for full-annulus and PSM ($M = 4$) solutions after 20 rotor revolutions. $G = 0.05, Re_\phi = 4.3 \times 10^4$	138
5.9. Comparison of non-dimensional relative angular phase velocity of the third spatial harmonic, $m = 3$, of the radial velocity between PSM ($M = 4$) and full-annulus solutions at a numerical probe located on the mid-cavity axial plane for the rotor-stator cavity. $Re_\phi = 4.3 \cdot 10^4$; $G = 0.05$	139
5.10. High-pressure aeroengine compressor and schematic diagram of open rotating cavity [4].	140
5.11. Computational mesh and main geometrical parameters of a generic compressor cavity, where $R_0 = 400\text{mm}$ and $R_i = 140\text{mm}$ and the gap between coaxial rotating disks is $s = 80\text{mm}$	141
5.12. Reduced-passage sample mesh of the compressor cavity with a sector pitch of 10°	142
5.13. Dimensionless unsteady radial velocity contours at mid-cavity compressor axial plane after 30 rotor revolutions.	143
5.14. Comparison of radial velocity contours on the mid-cavity compressor axial plane for full-annulus and PSM solutions after 30 rotor revolutions. $Re_\phi = 1.95 \times 10^6$	144
5.15. Comparison of non-dimensional relative angular phase velocity of the third spatial harmonic, $m = 3$, of the radial velocity between PSM ($M = 6$) and full-annulus solutions at a numerical probe located on the mid-cavity axial plane for the compressor cavity. $Re_\phi = 1.952 \cdot 10^6$	145
5.16. Computational mesh and main geometrical parameters of a generic turbine rim seal cavity, , where $R_0 = 150\text{mm}$ and $R_i = 40\text{mm}$, $s = 9\text{mm}$ and the axial gap pf the rim seal is $b = 3\text{mm}$	148
5.17. Reduced-passage 1/78th sector mesh of the rim seal cavity configuration. . . .	149
5.18. Schematic distribution of five j -th reduced-passage samples used by PSM with $M = 2$	150

5.19. Comparison of unsteady radial velocity contours at mid-axial plane of the turbine stage stator/rotor cavity for full-annulus and PSM solutions after 20 rotor revolutions.	151
5.20. Instantaneous static pressure field on the rotor wall of PSM ($M = 14$) solution after 20 rotor revolutions.	152
5.21. Comparison of ingestion flow rates between full-annulus and PSM ($M = 14$ and 6) solutions monitored at the numerical probe in Fig. 5.20.	153
A.1. Harmonic content for $k_L = 1$ and $k_c = 6$	181
B.1. Physical model of two coaxial rotating infinite disks.	184
B.2. Sketch of the flow pattern between two infinite disks.	186
B.3. Dimensionless radial (F) and circumferential (G) velocity distributions for a rotor-stator cavity rotating at $Re_s = 975$	191
B.4. Dimensionless velocity components (a) and temperature field components $\tilde{T}_0$ and $\tilde{T}_2$ (b) at $Re_s = 2,000$; $Pr = 0.71$; $Ec_r = 1$ and $\tilde{r} = 5$	193
B.5. Renormalised non-dimensional shear stress ($\tilde{\sigma}/Re_s^{1/2}$).	195
B.6. Renormalised non-dimensional heat flux balance ($qs/k(T_R - T_s)Re_s^{1/2}$).	196
B.7. Dimensionless velocity components at the midpoint of the core region ($\xi = 0.5$) as a function of dimensionless time scaled with the diffusive time at $Re_s = 2,000$ for a disk impulsively started from rest (middle). Velocity fields at; $\tau = 0.2Re_s$ (Stage II) and $\tau = 0.52Re_s$ (Stage III), left and right respectively.	200
B.8. Dimensionless time of Stage II and III for different initial conditions.	201
B.9. Transient evolution scaled with the diffusive time of the axial and circumferential dimensionless velocities (a) and dimensionless temperature (b) at the midpoint of the core region ($\xi = 0.5$) when the cavity is initially at rest ($G_0 = 0$ at $\tau = 0$) and when it is uniformly initialised ($G_0 = 10^{-4}$ at $\tau = 0$), at $Re_s = 4,000$; $Pr = 0.71$; $Ec_r = 1$	203

B.10. Transient evolution scaled with the diffusive time of the renormalised Nusselt number ($Nu/Re_s^{1/2}$) at the stator and rotor with $G_0 = 10^{-4}$, at $Re_s = 4,000$; $Pr = 0.71$; $Ec_r = 1$	204
B.11. Dimensionless temperature field with $G_0 = 0$ (a) and $G_0 = 10^{-4}$ (b) at different dimensionless times for the case $Re_s = 4,000$; $Pr = 0.71$; $Ec_r = 1$	205
B.12. Dimensionless axial, circumferential velocities and temperature at the midpoint of the core region ($\xi = 0.5$) as a function of dimensionless time scaled with the diffusive time at $Re_s = 4,000$ for a speed-down regime where the rotating disk is suddenly stopped (middle). Velocity fields at; $\tau = 0.02Re_s$ (Stage II) and $\tau = 0.1Re_s$ (Stage III), left and right respectively.	207
B.13. 1D-Analytical Model: Time evolution scaled with the convective time of dimensionless temperature at the core region ($\xi = 0.5$) for different dimensionless speed-up times. ($Re_{s,f} = 4,000$; $Pr = 0.71$; $Ec_{r,f} = 1$).	209
B.14. Influence of a stationary outer shroud on the steady laminar flow ($Re_s = 4,000$, $Re_\phi \simeq 10^5$) at the midpoint of the core region ($\xi = 0.5$) of a square cavity with a gap ratio $Y = 0.2$. (a) Dimensionless azimuthal velocity. (b) Dimensionless temperature. ($Pr = 0.71$, $Ec = 8 \times 10^{-2}$)	211
B.15. Dimensionless azimuthal velocity, $v_\theta/\Omega b$, and temperature, $\tilde{T}$, contour plots on a finite laminar rotor-stator cavity at two different dimensionless time; (a) and (c) $\tau = 4$; (b) and (d) $\tau = 22$. ($Re_s = 4,000$; $Pr = 0.71$; $Ec = 8 \times 10^{-2}$; $Y = 0.2$)	212
B.16. Transient evolution scaled with the convective time of the components of the energy balance of a rotor-stator cavity with gap ratio $Y = 0.2$, represented by Nusselt number based on the outer radius. Case of $Re_s = 4,000$; $Pr = 0.71$; $Ec = 8 \times 10^{-2}$	214
B.17. Comparison of the non-dimensional circumferential velocity profiles obtained from 2D numerical simulations (Solid symbols) and from the simplified similarity analysis (Filled symbols) at different time instants at the same dimensionless radius ($x = 0.45$).	215

B.18. Time evolution scaled with the convective time of the dimensionless temperature at the core region ($\xi = 0.5$) of a two-dimensional rotor-stator closed cavity of aspect ratio $Y = 0.2$ at a dimensionless radius $x = 0.4$ for different dimensionless speed-up times. ($Re_{s,f} = 4,000$; $Pr = 0.71$; $Ec_{r,f} = (\frac{x}{Y})^2 \cdot Ec_f = 0.32$). 216

List of Tables

3.1.	NASA rotor 67 design operating conditions.	65
3.2.	Computational cost of different numerical approximations for NASA rotor 67 case under distorted inflow conditions for a mesh of 2.27×10^6 nodes per passage.	71
3.3.	Computational time for the NASA rotor 67 unstable cases under rotating stall for a mesh of 3.53×10^6 nodes per passage.	90
5.1.	Computational cost of different numerical approximations for rim seal config- uration cavity for a mesh of 0.57×10^6 nodes per reduced-passage sample.	154
A.1.	Bessel functions values of the first kind $J_n(\beta)$ for modulation index cases in A.1.	182
B.1.	Constant values for the non-dimensional conservation of angular momentum.	194
B.2.	Constant values for the non-dimensional energy budget for $Pr = 0.71$	197

Nomenclature

Abbreviations

CFD	Computational Fluid Dynamics
FULL	Full-annulus
LWE	Linearised Wave Equation
ND	Nodal Diameter
OP	Operating Point
PR	Pressure Ratio
PSM	Passage-Spectral Method
URANS	Unsteady Reynolds-Averaged Navier-Stokes

Greek Symbols

β	Modulation index; core swirl ratio
ΔP	Total pressure amplitude
Δp	Static pressure amplitude
Δt	Time step
Δx	Spatial discretization

ε	Control factor of the non-uniformity mean flow velocity
γ	heat capacity ratio
λ	Characteristic wavelength
μ	Dynamic viscosity
ν	Kinematic viscosity
Ω	Rotational speed of the rotor; control volume
ω	Angular frequency; rotational speed of the fluid
Φ	Viscous dissipation term
ρ	Density
τ	Non-dimensional time; viscous stress tensor
θ	Circumferential coordinate
ξ	Non-dimensional axial distance
ζ	Local circumferential coordinate

Roman Symbols

$\dot{m}$	mass flow
$\hat{\mathbf{U}}$	Vector of complex Fourier coefficients
$\mathbf{D}$	Vector of artificial dissipation fluxes
$\mathbf{e}$	Unit vector
$\mathbf{F}$	Vector of inviscid and viscous fluxes
$\mathbf{S}$	Vector of source terms

$\mathbf{U}$	Vector of conservative variables
$\mathbf{v}$	Absolute velocity
$\mathbf{x}$	Cylindrical coordinate vector
$\tilde{r}$	Dimensionless radial distance
a	Speed of sound
c	Chord length; heat capacity
C_w	Non-dimensional cooling air mass flow
Ec	Eckert number
F	Dimensionless radial velocity
f	Frequency
G	Aspect ratio of the cavity; dimensionless circumferential velocity
H	Dimensionless axial velocity
h	Heat transfer coefficient; enthalpy
K	Block spacing location
k	Thermal conductivity; spatial harmonics
k_c	Embedded spatial harmonics
k_L	Long wavelength spatial harmonic
k_S	Short wavelength spatial harmonic
L	Characteristic length

M	Number of total spatial harmonics; Mach number
m	Spatial harmonic index
M_{is}	Isentropic Mach number
N	Number of periodic embedded harmonics inside each block
n	Time step level
N_B	Number of total blocks or passages
N_b	Number of rotor blades
N_H	Number of unsteady harmonics
N_J	Number of blocks retained in the Fourier transform
N_P	Number of nodes inside each block
N_v	Number of vanes
Nu	Nusselt number
P	Total pressure; Block-Jacobi preconditioning matrix; pitch
p	Static pressure
Pr	Prandtl number
Q	Volumetric heat sources
q	Convective heat flux
R	Speed-up factor; the gas constant; disk radius
Re	Reynolds number

Re_ϕ	Rotational Reynolds number
Re_s	Reynolds number based on the axial gap
s	Gap between disks
St	Reduced frequency
T	Temperature; time period
t	Time
x, y, θ	Cylindrical coordinates
Y	Cavity gap ratio

Subscripts

i	Node index
j	Block or passage index
m	Spatial harmonic index
n	Time level step

1. Introduction

Contents

1.1. Background and Motivation	2
1.2. Numerical Methods Challenges in Turbomachinery	7
1.3. Objectives and Outline	21

Due to relative rotation of blade rows, the flow in aeroengines and gas turbines is inherently unsteady. This can lead to the occurrence of an efficiency loss and ultimately engine failure. Besides, the blading forces the flow to turn through multiple identical blade passages in each blade row with the ultimate goal of extracting or injecting work. This function-driven feature leads to a common and fundamental characteristic of flow disturbances in turbomachinery: the spatial and temporal periodicity. In space, a general periodicity certainly holds for a domain covering the whole annulus. The geometrically identical blade passages in each blade row are designed to behave in the same manner in space and in time. Although instantaneous flow in the different blade passages within each blade row is not identical in practice, it is not expected to differ abruptly from one passage to another. This is true for most of the flow conditions of practical interest. However, the flow, in general, is three-dimensional and unsteady with a wide range of gas velocity conditions and subject to complex geometries.

Understanding and predicting unsteady aerodynamics is crucial to engine design. It is recognised, however, that there is always a need for developing efficient methods for applications to blade designs. This basic requirement for fast prediction methods in a multi-disciplinary design environment remains unchanged, regardless of computer speed. Computational efficient

methods allow the designer to test and optimise a wide range of conditions and geometries at a relatively low cost. As sometimes at operating conditions experimental techniques are either not feasible or have a limited accuracy, with the ultimate goal to predict flow behaviour in modern aero-engines, as illustrated in Fig. 1.1 for a real engine, without simulating the entire domain. Therefore, in many cases, numerical simulations are the only way to understand complex flow physics systematically. Recently, there has been a considerable progress in the development of Fourier methods for turbomachinery applications. The main driver is to develop efficient and accurate computational methodologies for the prediction and analysis of unsteady effects in turbomachinery. Flow unsteadiness is associated with the rotational speed or the natural vibration frequency of the blades. The frequency of this kind of unsteadiness is known beforehand and is exploited by tailored numerical methods. The ability to accurately model unsteady flows is crucial to safety and performance. In that way, there have been considerable efforts devoted to the development of novel numerical techniques in Computational Fluid Dynamics (CFD) which, making assumptions about the time/space periodicity of the flow, take advantage of it to reduce the computational cost required to obtain the desired solution. This problem is exacerbated when dealing with unsteady, multi-row configurations. Frequently, the flow is assumed to be periodic in time with a known fundamental frequency associated with the forcing of the flow. This is actually the case in forced response or uncoupled flutter problems where the fundamental frequencies are respectively, the blade-passing frequency and the natural frequency of the rotor blade.

1.1. Background and Motivation

The general property of the blade passage-to-passage periodicity has had an overwhelming influence on the development of simulation methods in the past. The circumferential truncation from the whole annulus with multiple blade passages to a single blade passage domain has been almost invariably used in all the prediction and analysis methods for conventional blading design systems. A phase-shift periodicity is formed on the basis that adjacent blade passages

Figure 1.1.: Perspective view of the Rolls-Royce UltraFan aero-engine.

should see the disturbances of the same amplitude but with a constant temporal phase angle difference. Periodically unsteady flows occur as a result of the rotation through a non-uniform flow field and can thus be caused by any flow distortion. Unsteadiness in turbomachines is not necessarily correlated to the wheel speed of the engine or structural eigenfrequencies. Vortex shedding or global instabilities such as surge (characterised by large amplitude pressure oscillations) introduce distinctive frequencies which are not related to rotation. Some of these complex unsteady and non-axisymmetric flows are described below and used as the application target of the proposed method described in this thesis.

1.1.1. Unsteady Flows in Fan-stages

Inlet flow conditions are distorted when the aircraft is flying at an angle of attack or in cross wind conditions. Flow at a large incidence creates regions of high and low momentum fluid in the intake. The presence of pylons or air frame boundary layers in integrated engines can also give rise to non-uniform intake flows [5, 6]. Inlet distortions tend to have a long wave length and can propagate a long distance with little decay. Unsteady flow modeling is also needed when studying the influence of inlet distortions on compressor stability. Under these circumstances

Figure 1.2.: Instantaneous entropy contour after stall inception.

the fan and the downstream rotors will move in and out of low total pressure zones. The exposure time of rotors to this distorted zone and its extent seem key interrelated stability factors. In open rotor engines, the unsteady wakes from upstream rotors can strongly influence the flow in the intake ducts [7, 8]. This rotor-duct-compressor system in itself gives rise to a large-scale unsteady coupled system. However, surge is preceded by stall. Rotating stall or surge are characterised by a complete breakdown of the steady flow through the whole compressor and the subsequent aerodynamic excitation of the stage in which the created circumferential flow pattern, as sketched in Fig. 1.2 for a rotor row under rotating stall conditions. Avoiding stall inception in operation conditions is one of the most important issues for a fan designer. From the performance point of view, it is desirable to operate the fan at the highest pressure ratio and efficiency but such operating conditions are inherently unstable due to aerodynamic instability phenomena.

Since the performances of an engine fan are known to be highly impacted by inlet distortion, it is important to develop a fast and efficient numerical tool which can accurately and efficiently predict the performance of a fan subjected to large pitch inlet distortion during the design process. The complex nature of unsteady flows in a fan-stage, with clean or distorted inflow conditions, presents then a computational and numerical challenge as full-annulus simulations and long

domains are required to accurately capture these instabilities [9, 10, 11, 12]. Computationally efficient analysis tools for this kind of stall phenomenon when designing rotor blades are highly desirable. The formation of stall inception is a local instability in which the circumferential flow pattern is disturbed and reduced flow rate gives rise to flow separation, results in large fluctuations of flow with respect to time where one or more stall cells propagates at an unknown fraction of the rotor speed. The asymmetry of the flow causes non-uniform circumferential pressure distortions which can trigger a strong aeromechanical response in the fan blades. This kind of operating conditions at the fan is usually carried on by expensive unsteady full-annulus calculation due to the fact that numerical analysis considering only a part of annulus may not predict accurately the unsteady features of the long length scale disturbances flow due to rotating stall as well as the short length disturbances of the order of the blade pitch with high nonlinearities. It was observed that both could exist in the same compressor and could lead to a rotating stall [13, 14]. This phenomenon was previously investigated by some researchers as stall inception [15, 16]. Experimental works related to the stall process due to the influence of distorted inflow configurations have been previously reported by other researchers [17, 18, 19]. It is then necessary to accurately predict the stability limit to stall inception at peak pressure rise condition [20], on the order of one rotor circumference with the initiation of rotating stall, which emerged due to tip clearance vortex breakdown [21]. Computationally efficient tools for the analysis and design of fans and compressors able to capture this kind of phenomena are highly desirable. The ability to describe and quantify the flow perturbations as they progress through the fan and the compressor when subject to inlet distortion is very important as well.

1.1.2. Rotating Instabilities in Cavities

In modern aero-engines approximately ten percent of the main annulus gas is bled off to feed cooling and sealing systems. Their optimization is considered one of the most promising techniques to increase global engine efficiency. The accurate prediction of the stator, rotor and disk metal temperature is crucial for the reliable layout of such air systems, in order to avoid that the gas temperature can exceed the metal's melting point. In order to increase the specific power of

Figure 1.3.: Schematic diagram of non-axisymmetric flow structures in a rotating cavity core.

gas turbines and consequently reduce the size of the power plant, higher turbine entry temperatures of the main gas flow, accompanied by higher pressure ratios are required. Components in contact with such a high temperature flows rapidly exceed their creep and fatigue limits, leading to catastrophic failure. The operation at these elevated temperatures is only possible because of the internal air system, which uses some of the compressor's air to cool the turbine disks, blades and nozzle guide vanes. The amount of cooling air needs to be minimized so as not to adversely affect turbine efficiency. The air used for cooling cavities will be heated as a result of viscous dissipation as it flows over both rotating and stationary surfaces. As the flow passes the vanes and the blades in the annulus, a pressure gradient is formed radially outward through the rim seal at different circumferential locations. The existence of large scale unsteady flow structures manifested in contra-rotating vortex pairs unrelated to blade passing has been previously identified in rotor disk cavities [22, 23]. These rotating low pressure regions can reduce the lifetime of the turbine by overheating the components, induced by ingestion of hot gas, and influencing the mechanical integrity of the device. The resultant general characteristics are that the flow into a rotor-stator cavity is both unsteady and non-axisymmetric in a largely or completely axisymmetric configuration. Typically, these unsteady structures is associated with low frequencies and accompanied by higher frequency fluctuations caused by perturbations with a length-scale of the order of the blade passage pitch [24, 25, 26, 27]. Better understanding of the flow structure can aid to improve the turbine efficiency.

In general, rotating cavities are formed between adjacent rotating and non-rotating disks and a shroud. Apart from an externally imposed three-dimensionality, the flow in a cavity can become

three-dimensional due to the formation of vortex structures. In a rotor-rotor compressor cavity, fluctuating pressure structures, as shown in Fig. 1.3 in the inner part of the cavity, which are previously shown to form inside the rotating disk cavity due to buoyancy-induced flow [28], can affect the heat transfer. The heat transfer from the disk depends on the disk temperature, and the temperature depends on the heat transfer. Under this point of view, a non-axisymmetric radial flow movement in the core region of the cavity may be triggered by circumferential geometrical and flow distortions which exist in reality. The resultant general characteristics are that the flow into a rotating cavity is both unsteady and non-axisymmetric in a largely or completely axisymmetric configuration.

Elementary models that rely on empirical validation and are currently used in design do not account for some of the known flow mechanisms, and prediction of sealing performance with CFD has proved challenging [29, 30, 31]. Numerical and experimental studies have indicated important unsteady flow effects that explain some of the differences identified in comparing predicted and measure sealing effectiveness, many of the experimental and computational studies that have identified non-blade passing related unsteadiness believed to be associated with rotating flow modes [32, 33, 34]. With the demand of further aerothermal performance enhancements there is an increasingly need for better understanding of the dominant interaction mechanisms and influencing parameters. The non-axisymmetric nature of the flow into a rotating cavity with a number of counter-rotating vortex pairs presents a computational challenge as a full 360° circumferential domain will be needed, requiring a significant amount of computational resource. The accurate prediction of these complex instabilities at the inviscid core region is crucial for the reliable layout of such air systems, but also adequate analysis tools for this kind of rotating cavity flows in a design process.

1.2. Numerical Methods Challenges in Turbomachinery

The rapid growth of computer power in addition to the advancements in numerical techniques have made CFD methods a powerful and essential tool for modelling turbomachinery flows.

This development of computer power has made possible to simulate direct non-linear solutions of flow in a whole annulus domain, although the computing resources required are also increased significantly in proportion. Compared to a steady flow solution a direct unsteady Reynolds-averaged Navier-Stokes (URANS) solution will typically require about 3-4 orders of magnitudes more computing time. CFD solutions in combination with theoretical approaches can improve gas turbine design by offering a powerful way to develop a physical understanding of the flow phenomena in complex systems. Modeling unsteady aerodynamic effects using full-annulus transient simulations with multiple blade rows is feasible [35, 36], but is computationally expensive when using time-marching methods. The computational time spans over several engine revolutions and long computational times with large meshes are required for an accurate solution. The fundamental problem in the context of an axial turbomachine stage is the lack of geometric periodicity at a passage level when rotor and stator have different blade counts. It is needed then to develop efficient and accurate numerical methods to simulate non-axisymmetric and unsteady flows that occurs in a whole annulus domain.

In general, Fourier series can be used to approximate periodic nonlinear flow disturbances both in space and in time. The fundamental modelling requirement is the existence of periodicity in a certain direction, which as mentioned earlier, is one of the distinct characteristics of turbomachinery flows. Discrete Fourier series is a powerful approach for the efficient simulation of nonlinear flow disturbances in turbomachinery. The unsteadiness can arise from a variety of sources and can be either periodic or non-periodic. The Fourier model, by its very nature, is a truncated (reduced) model, which does not solve all the length scales as a direct method (mesh-resolution permitting). The overriding consideration is a clear appreciation of the relevant length scales for the problems to be solved. The filtering (truncating) ability of a Fourier model should be regarded as a strength rather than as a limitation for practical turbomachinery problems.

Extensive reviews of Fourier based methods in turbomachinery have been given by He [37], and Dowell and Hall [38], where a daunting array of modeling and numerical methods and strategies can be found. The numerical methods discussed in the following lines represent, to the author's

knowledge, the state-of-the-art in the prediction of unsteady and non-axisymmetric flows in turbomachinery. Nowadays, modern computational techniques for solving time periodic flows in turbomachinery can be divided in: nonlinear time-marching [39, 40], linearised frequency domain [41, 42, 43, 44] and non-linear frequency domain methods [45, 46]. Each has their own limitations, but researchers have demonstrated that it is possible to accurately predict unsteady and non-axisymmetric flows in turbomachinery under certain conditions. The reviewed work suggests that CFD, as ever, is an activity that needs a thorough revision of the processes and tools, and requires overseeing of the modeling practices.

1.2.1. Linear Frequency-domain Methods

Frequency domain linearised Navier-Stokes solvers methods rely on the assumption that unsteady perturbations are periodic in time and small compared to an underlying steady-state flow, assumptions which are violated by most compressor and turbine blades. Nevertheless, the techniques developed provided useful insights into the physics of turbomachinery flow and were used for aerodynamic design [47]. To linearise the equations, each variable is expressed as a mean value $\bar{\mathbf{U}}$ and a small perturbation $\mathbf{U}'$ about that value

$$\mathbf{U}(\mathbf{x}, t) = \bar{\mathbf{U}}(\mathbf{x}) + \mathbf{U}'(\mathbf{x}, t) \quad (1.1)$$

being

$$\mathbf{U}'(\mathbf{x}, t) \ll \bar{\mathbf{U}}(\mathbf{x}, \theta) \quad (1.2)$$

where $\mathbf{x} = (x, r, \theta)$ describes the coordinate of every node inside the discretised domain at which the solution is computed. The steady part is obtained from a steady state calculation and the unsteady part is decomposed into its Fourier components, $\hat{\mathbf{U}}_n$, which are solved separately from the steady flow.

$$\mathbf{U}'(\mathbf{x}, t) = \sum_{n=1}^{N_H} \hat{\mathbf{U}}_n(\mathbf{x}) e^{I \omega_n t} \quad (1.3)$$

With the assumption of a periodic and small perturbations, the time-derivative can be eliminated

and the problem becomes linear and steady with variable coefficients depending only on the steady-state solution. In order to include wakes, boundary layers or secondary flows, researchers developed time-linearised solvers for the Reynolds-Averaged Navier- Stokes equations with turbulence models to model the Reynolds stress terms [48, 49, 50]. These methods offer a good compromise between computational efficiency and flow accuracy in the absence of highly non-linear phenomena such as strong shock waves or stall conditions. However, even these cases the linear theory shows that, given a sufficiently fine grid, the calculations produces the correct unsteady pressure on the blades for an unloaded flat-plate cascade [51]. Another example is given by the time linearized solver $Mu^2s^2T - L$, developed at ITP Aero, where a multiple row coupling mechanism has been implemented inside the in-house CFD solver Mu^2s^2T for the 3D RANS equations. Its capabilities to perform unsteady simulations in turbomachinery have been validated for flutter [44, 52] as well as for turbine tone noise prediction [53, 54, 55].

1.2.2. Non-linear Frequency-domain Methods

Since the temporal harmonics are solved independently, the non-linear interactions between different frequencies or between the unsteadiness and the mean flow are neglected. When the unsteadiness of the flow is driven by flow instabilities the fundamental frequency is not *a priori* known and needs to be determined as part of the solution. Still, some methods have been attempted to expand the problem in a time Fourier series [56], but the success and the practical application of these methods have been limited. Harmonic methods which incorporate some of these non-linear effects and multi-row coupling are briefly described below.

1.2.2.1. Non-linear Harmonic Method

The non-linear harmonic method developed by Ning and He [42] is based on a sequence of frequency domain linear analyses. The baseline flow is the resulting time-averaged flow instead of a steady state flow [57]. The time averaging of the non-linear flow equations results in additional terms in the momentum and energy equations. These deterministic stresses account for the non-linear effects of the unsteadiness on the time-averaged flow [58] and are referred to

as unsteady or deterministic stress terms because they look similar to the Reynolds stress terms. Since the two equations are steady, they can be marched to the steady-state in pseudo time like conventional time-linearised methods. However, because the deterministic stress terms render the time-averaged and the perturbation equations interdependent, the coupled system needs to be integrated simultaneously in time.

Non-linear harmonic methods offer advantages over linear harmonic methods since retain interactions between the perturbations and the mean flow, which are mainly visible in regions with shocks where conventional time-linearised methods fail to catch the smearing effects of non-linearities and predict sharp pressure peaks. However, non-linearities arising from the interaction between different disturbances are still neglected.

1.2.2.2. Harmonic Balance Method

Further frequency methods include the ‘harmonic balance technique’ introduced by Hall [45] for cascade flows. This method derives the temporal behaviour of the solution from uniformly sampled time instants. For an analysis including N_H unsteady harmonics, steady state solutions at $2N_H + 1$ time levels are computed. The solutions are linked through phase-lagged boundary conditions and a spectral time-derivative operator in the interior domain. From the knowledge of the temporal behaviour in time, the temporal Fourier coefficients can be constructed. While non-linear frequency methods only contain the non-linear interaction between the time-averaged flow and the first temporal harmonics, the harmonic balance methods can also include higher order harmonics. This offers advantages but is comparatively expensive since each harmonic solution requires two steady state calculations (real and imaginary part) and all harmonic solutions must be computed at the same time. The computational time therefore scales with the number of harmonics to be included in the solution. Harmonic methods which incorporate some of these non-linear effects and multi-row coupling has been extensively developed in the recent years [46, 59, 60]. Since the harmonic components are solved independently, potentially important non-linear interactions between different frequencies or different kind of unsteadiness and the mean-flow are neglected.

Figure 1.4.: Single passage domain with periodic boundaries.

1.2.3. Non-linear Time-domain Methods

All axial-flow turbomachines have, to some degree, azimuthal periodicity. In a zero-order approximation, the performance of a component (e.g., a rotor) can be calculated by assuming azimuthal periodicity and so one single passage can be simulated to compute the performance of the entire component. However, most turbomachinery stages have different pitch counts between rotor and stator, i.e., different number of blades of the rotor in comparison to the stator, in order to avoid tonal noise, reduce weight, etc. One is then forced to simulate several passages of rotor/stator in order to compute component performance. Because of the additional cost of simulating multiple passages (which can make the whole simulation unfeasible depending on the blade count) different techniques can be applied so that one ends up simulating one single passage. These techniques imply some sort of approximation.

1.2.3.1. Periodic Single-passage

When the problem's geometry allows it, the computational domain can be truncated to a periodic sector. Periodic boundary conditions can then be applied to the pitchwise boundaries of the first and last passage in each row. However, such a periodic sector only exists if all blade

rows have a common factor which becomes unlikely when multiple stages are considered. Furthermore, even if a periodic sector can be identified, it is not guaranteed that the circumferential wavelength of the unsteady disturbances in the domain also coincides with the periodic sector. The periodicity between both periodic boundaries “*per-up*” and “*per-down*” can be expressed as:

$$\mathbf{U}_{Per-up}(\mathbf{x}, t) = \mathbf{U}_{Per-down}(\mathbf{x}, t) \quad (1.4)$$

The flow equations in the single passage domain are still solved by the time-marching (see Fig. 1.4). The boundary conditions at the periodic boundaries of a single-passage domain require relation between the flow variables at both two boundaries at the same time level.

1.2.3.2. Phase-lagged Boundary Conditions

The whole annulus approach remains computationally expensive and is not practical in the design process. Over the past few decades time-domain single passage methods have become available which offer considerable savings compared to whole annulus computations. With an appropriate representation of the phase-lagged periodic boundary conditions, it becomes possible to reduce the computational domain to one passage per row. In order to implement the pitchwise boundary conditions, these single passage methods use additional ‘dummy’ points which extend into the domain of the adjacent passage (see Fig. 1.4). Each shadow point corresponds to a master point inside the single passage domain. By collecting information about the temporal variation on the master points, the flow variables at the shadow points, which form the pitch-wise boundary conditions, can be determined. The phase-lagged periodicity is expressed as:

$$\mathbf{U}_{Per-up}(\mathbf{x}, t) = \mathbf{U}_{Per-down}(\mathbf{x}, t \pm |\Delta T|) \quad (1.5)$$

where the temporal phase shift in the rotor $|\Delta T_R|$ is given by:

$$|\Delta T_R| = \frac{2\pi}{\Omega N_b} \quad (1.6)$$

where Ω is the rotational speed in rad/s and N_b is the number of rotor blades. Similarly for the stator:

$$|\Delta T_s| = \frac{2\pi}{\Omega N_v} \quad (1.7)$$

where N_v is the number of vanes. The sign of the phase shift in Eqn. 1.4 depends on the direction of the relative rotational speed of the two blade rows. If the disturbance is travelling in the direction of positive θ the left boundary lags the right boundary and vice-versa.

1.2.3.3. Time-inclined Method

Giles [40] modelled stator-rotor interaction using a ‘time inclination method’. In this method, the equations are transformed in time so that, in the transformed domain, the solution is periodic in space and time (corochronic periodicity). Giles was the first to realize that one can recover temporal periodicity by transforming the physical coordinates (x, y, t) to the computational coordinates (x', y', t') , being $t' = t - \lambda y$, where $\lambda = \Delta T / P$ is the ratio between the time lag between periodic boundaries and pitch, P . This transformation must, then, be applied to the governing equations. In the computational plane t' is constant at each time step. In [40], Giles elegantly showed that the relationship between transformed and physical variables is exact when the Euler equations are transformed. In multi-row calculations, the stator and rotor grids require different values of the transformation parameter λ since their pitches are different but the time step size is the same for both rows. If the equations are transformed and solved in the new coordinate system, the stator/rotor pitch ratio becomes unity and the phase lagged periodicity will automatically be satisfied. This approach does not assume temporal periodicity and is thought to offer advantages in viscous flows, where physical instabilities such as vortex shedding induce frequencies which are not a multiple of the blade-passing frequency. However, due to stability requirements the method is restricted to certain blade count ratios. The time inclination parameter λ must lie in the domain of dependence of the physical solution.

1.2.4. Non-linear Methods for Non-axisymmetric Flows

The methods described in the previous section were developed for unsteady axisymmetric flows. Limitations apply to the representation of non-axisymmetric configurations and non-linear interactions where the associated frequencies are not known. Although the major developments have been focused on the temporal Fourier harmonic modelling, recent progress in the use of spatial Fourier modelling is also described. The methods described up to now concern the modelling of timewise variations. However, Fourier modelling can also be used in space to reduce computational resources, as long as a spatial periodicity can be defined. The establishing of circumferential periodicity for unsteady flows associated with aerodynamic interactions has lent the problem to application of this Fourier model. The spatial Fourier series is used in truncating the full annulus to a subset of single blade passage domains. The flow equations in the single passage domain are still solved by the time-marching, whilst the Fourier series is used at the blade-to-blade periodic boundaries (as shown previously in Fig. 1.4). The boundary conditions at the periodic boundaries of a single-passage domain require relation between the flow variables at both two boundaries.

1.2.4.1. Spatial Modelling Using Discrete Fourier Series

Turbomachinery flows are characterised by their spatial cyclic symmetry under certain conditions, and numerical methods taking advantage of it have been proposed to reduce their complexity and cost. The spatial variation of the flow variables along the wheel circumference can be approximated then by a discrete Fourier series to continuously correct each other according to the spatial periodicity for axisymmetric geometries. He [61] pioneered the modelling non-axisymmetric steady and unsteady flows using time-domain Fourier methods. The purpose of using the spatial Fourier approximation is to reduce the computing resources required by a direct calculation of a full circumference domain. The full annulus can be truncated to such an extent as long as it can retain the correct information to carry out the Fourier transform.

To approximate sinusoidal spatial variations using a discrete Fourier series with the three unknown variables: the mean value, $\bar{U}$, and two Fourier coefficients, A_m and B_m , need to be com-

Figure 1.5.: Computational mesh cell and its circumferential neighbours points.

puted at a minimum. In this case the flow field $\mathbf{U}$ can be represented as $\mathbf{U}(\mathbf{x}, t) = \bar{\mathbf{U}}(x, r, t) + \mathbf{A}_m(t) \sin(k\theta) + \mathbf{B}_m(t) \cos(k\theta)$, where m is the index of the m -th Fourier coefficient and M is the total number of the retained spatial harmonics in the discrete Fourier series. This turns into a system of three equations with three unknown variables. This system is solved for each of the new nodes at both periodic boundaries (see Fig. 1.4), where the solution needs to be updated taking into account the spatial variation of the flow variables along the wheel circumference. For the first spatial harmonic, only three uniformly spaced points are needed to represent exactly a sinusoidal function. Three samples will be fit exactly by a 3-term Fourier series. In general, you can represent a number of M spatial harmonics with a number of $2M + 1$ samples into your Fourier series.

Consider now a circumferential non-uniform flow with a wavelength of the whole circumference. Taking a cylindrical coordinate system (x, r, θ, t) , we approximate the circumferential variation by the m -th order discrete Fourier series, thus

$$\mathbf{U}(x, r, \theta, t) = \bar{\mathbf{U}}(x, r, t) + \sum_{m=1}^M [A_m(x, r, t) \cdot \sin(m\theta) + B_m(x, r, t) \cdot \cos(m\theta)] \quad (1.8)$$

where $\bar{\mathbf{U}}$ is the circumferentially averaged value of flow variable $\mathbf{U}$. The Fourier coefficients A_m and B_m only depend on the axial and radial coordinates (x, r) . To determine the three unknown Fourier coefficients $\bar{\mathbf{U}}$, A_m and B_m , we evaluate Eqn. 1.8 at the $2M + 1$ uniformly spaced samples for which we have flow field solution. Spatial Fourier coefficients are then obtained by

summation of the flow variables at the uniformly spaced circumferential positions j :

$$\begin{aligned}\bar{\mathbf{U}}(x, r, t) &= \frac{1}{2M+1} \sum_{j=1}^{2M+1} \mathbf{U}_j \\ A_m(x, r, t) &= \frac{2}{2M+1} \sum_{j=1}^{2M+1} \mathbf{U}_j \sin(m\theta_j) \\ B_m(x, r, t) &= \frac{2}{2M+1} \sum_{j=1}^{2M+1} \mathbf{U}_j \cos(m\theta_j)\end{aligned}\quad (1.9)$$

The sampling points j are uniformly spaced points along the circumference, where

$$\theta_j = \frac{j-1}{2M+1} 2\pi \quad (1.10)$$

The Fourier coefficients are time-dependent and need to be evaluated at every time step of the time marching scheme. Once Fourier coefficients have been calculated at each mesh cell, we need to evaluate/assign the flow variables at the dummy points to enable fluxes or differences calculations at each time-step iteration. If the phase for the mesh cell centre is $\Delta\theta$ (see Fig. 1.5), the flow values at the two dummy points are

$$\begin{aligned}\mathbf{U}(x, r, \theta_j + \Delta\theta, t) &= \bar{\mathbf{U}}(x, r, t) + \sum_{m=1}^M \{A_m(x, r, t) \cdot \sin[m(\theta_j + \Delta\theta)] + B_m(x, r, t) \cdot \cos[m(\theta_j + \Delta\theta)]\} \\ \mathbf{U}(x, r, \theta_j - \Delta\theta, t) &= \bar{\mathbf{U}}(x, r, t) + \sum_{m=1}^M \{A_m(x, r, t) \cdot \sin[m(\theta_j - \Delta\theta)] + B_m(x, r, t) \cdot \cos[m(\theta_j - \Delta\theta)]\}\end{aligned}\quad (1.11)$$

The resulting Fourier series gives the simultaneous and complete circumferential flow distribution.

Many problems of interest are characterized by two distinctive separate scales and a large multiplicity of similar small-scale elements. In a block-spectral approach, the global domain is decomposed into a large number of similar small blocks making use of scale-dependent solvability. The use of spatial Fourier methods for the computation of nonlinear flows in turbomachinery was first proposed by He [61]. In his original work, He used a block decomposition to study the steady 2D interaction between a fan OGV and a Pylon. Moreover, simplified unsteady cases with two-dimensional [61] or axisymmetric rotor disk cavities [62] were expanded in spatial Fourier series and subject to long-wavelength unsteady perturbations showing good

Figure 1.6.: Computational domain for circumferential Fourier model [1].

agreement between the spectral method and the full annulus solutions with significant computational savings. He applied the same idea to film cooling configurations [63], dimples [64] and lined acoustic intakes [65]. Recently, He [66] also extended the methodology to unsteady flows of short temporal and spatial scales (e.g. those due to self-excited unsteady vortices and turbulence disturbances), where a source term-based approach was adopted to facilitate a two-way coupling in terms of time-averaged flow solutions. Although the method could reproduce unknown frequencies caused by rotating instabilities the Fourier modelling was limited to an axisymmetric geometry. The work presented in this thesis exploits the same idea extending and applying the spatial Fourier modeling to more complex three-dimensional non-axisymmetric geometries, such as including blades in the reduced-passage domain.

1.2.4.2. Time-domain Fourier Approach

More recently, time-domain Fourier methods have been used for general non-axisymmetric flows across multiple blade rows by Stapelfeldt and Di Mare [67], who extended the method exploiting the time/space periodicity of the flow and Fourier modelling circumferential spatial variations within the same spatial block. The approach uses several blocks or reduced-passages

samples which are uniformly spaced across the circumference as shown in Fig. 1.6. A typical multi-row turbomachinery flow field is modelled on a reduced number of passages representing both travelling and stationary circumferential perturbations with N_P number of passages in the full wheel circumference. The passages are uniformly spaced across one circumferential wave length of the fundamental stationary disturbance and thus act as sampling points for a circumferential Fourier transform. The methodology relies as well on temporal Fourier transforms for the representation of the unsteady perturbations, and phase-lagged boundary conditions.

The time-domain Fourier method then assumes that the flow variables inside the single passage domain can be expressed as a superposition of a space-time average, stationary circumferential (passage-to-passage) variations and unsteady disturbances.

$$\mathbf{U}(x, r, \theta, t) = \bar{\mathbf{U}}_0(x, r) + \tilde{\mathbf{U}}(x, r, \theta) + \mathbf{U}'(x, r, \theta, t) \quad (1.12)$$

where $\bar{\mathbf{U}}_0$ is the space-time average, $\tilde{\mathbf{U}}$ is the stationary variation and $\mathbf{U}'$ is the unsteady fluctuation, which vary in amplitude and phase inside the single passage domain due to interaction with the steady flow field.

The method assumes that the unsteady fluctuations are travelling waves with constant angular velocity v and angular frequency ω . These requirements are fulfilled for blade row interaction problems, where the frequency and wavelength are determined by the rotational speed and blade count of adjacent blade rows. If more than one of these perturbations are present, the unsteady fluctuations can be described as a superposition of several independent disturbances where its frequency is *a priori* known. The flow variation using its temporal and space Fourier components is approximated as:

$$\begin{aligned} \mathbf{U}(x, r, \theta, t) = \sum_{n=1}^{N_N} \sum_{m=1}^M & A_{nm}(x, r) \cos(m\theta) \cos\left(\frac{\omega_n}{v_n} \theta - \omega_n t\right) + \\ & B_{nm}(x, r) \sin(m\theta) \cos\left(\frac{\omega_n}{v_n} \theta - \omega_n t\right) + \\ & C_{nm}(x, r) \cos(m\theta) \sin\left(\frac{\omega_n}{v_n} \theta - \omega_n t\right) + \\ & D_{nm}(x, r) \sin(m\theta) \sin\left(\frac{\omega_n}{v_n} \theta - \omega_n t\right) \end{aligned} \quad (1.13)$$

where N_N is the number of the superimposed disturbances. The spatial and temporal Fourier coefficients are obtained by summation over the $2M + 1$ uniformly spaced passages whereby the origin of the j -th passage is located at θ_j (previously defined in Eqn. 1.10):

$$\begin{aligned}
 A_{nm}(x, r)|_i &= A_{nm}(x, r)|_i \\
 &+ \frac{1}{N_T(2M+1)} \sum_{j=1}^{2M+1} [\mathbf{U}(x, r, \theta_j, t_i) - \mathbf{U}^*(x, r, \theta_j, t_i)] \cos\left(\frac{\omega_n}{v_n} \theta_j - \omega_n t\right) \cos(m\theta_j) \\
 B_{nm}(x, r)|_i &= B_{nm}(x, r)|_i \\
 &+ \frac{1}{N_T(2M+1)} \sum_{j=1}^{2M+1} [\mathbf{U}(x, r, \theta_j, t_i) - \mathbf{U}^*(x, r, \theta_j, t_i)] \cos\left(\frac{\omega_n}{v_n} \theta_j - \omega_n t\right) \sin(m\theta_j) \\
 C_{nm}(x, r)|_i &= C_{nm}(x, r)|_i \\
 &+ \frac{1}{N_T(2M+1)} \sum_{j=1}^{2M+1} [\mathbf{U}(x, r, \theta_j, t_i) - \mathbf{U}^*(x, r, \theta_j, t_i)] \sin\left(\frac{\omega_n}{v_n} \theta_j - \omega_n t\right) \cos(m\theta_j) \\
 D_{nm}(x, r)|_i &= D_{nm}(x, r)|_i \\
 &+ \frac{1}{N_T(2M+1)} \sum_{j=1}^{2M+1} [\mathbf{U}(x, r, \theta_j, t_i) - \mathbf{U}^*(x, r, \theta_j, t_i)] \sin\left(\frac{\omega_n}{v_n} \theta_j - \omega_n t\right) \sin(m\theta_j)
 \end{aligned} \tag{1.14}$$

where in the discrete case N_T is the number of equally discretised time steps per period of the travelling disturbance and $t_i = i\Delta t$. $\mathbf{U}^*(x, r, \theta, t_i)$ are the flow variables at the time step i as evaluated from the coefficients of time step $i - 1$. With the Fourier coefficients from Eqn. 1.14, the flow variables at time step i on the opposite periodic boundary, as shown in Fig. 1.4, are updated with the correct phase lag having into account the pitch of the single passage domain. The reconstruction using the Fourier transform ensures the temporal periodicity with the corresponding natural frequencies.

It is important to note that the existence of a chorochronic (space-time) periodicity is the only condition necessary for time-accurate Fourier methods such as the spatial Fourier method. Any unsteadiness must take the form of a travelling wave since the representation of unsteady disturbances is based on phase-lagged boundary conditions, where the frequencies and wave numbers of the disturbances must be known. Stapefeltd and Di Mare's method [67] extended Fourier to non-axisymmetric and multi-row geometries but this approach cannot solve unknown frequencies such as rotating stall or instabilities in cavities, as done by He's Fourier model [61, 62] for an axisymmetric domain.

1.3. Objectives and Outline

The present thesis is aimed at extending and applying the spatial Fourier modeling to 3D non-axisymmetric unsteady flows exhibiting long wave-length flow instabilities without any implicit assumption about the temporal periodicity of the flow. This approach gives the user control over the number of spatial harmonics employed to capture flow length scales spanning over several blade passages, significantly reducing the computational resources, removing harmonics that may be of no interest.

Frequency domain and single passage methods are typically limited to assumed periodically unsteady perturbations and ignore potentially important unknown variations in the underlying unsteady flow field. With regard the representation of unsteady instabilities, the proposed Passage-Spectral Method offers advantages over frequency-domain methods, since it makes no assumptions about the temporal periodicity of the flow and is able to reproduce a wide range of spatial harmonics with a significant reduction in the number of block-passage to simulate. The block-wise spatial Fourier model is well-suited for any flow which can be naturally decomposed into a finite number of circumferential Fourier modes. As passages can be placed at arbitrary circumferential locations, the size of the reduced passage model is independent of the blade count and is merely determined by the number of spatial harmonics to be retained in the Fourier approximation. This kind of methodology is applicable to the simulation of the onset of compressor instabilities (rotating stall) and the long wave-length instabilities which can be seen in the drum and under-platform cavities of turbines and compressors. The main uncertainty in applying this is its ability to trigger flow instabilities when the number of harmonics in the simulation is truncated. This fact can preclude the development of the excitation of the instability.

The objective of this research is to develop efficient numerical technique relevant to the study of rotating instabilities in complex unsteady flows for practical engineering applications without making any hypothesis about its temporal periodicity. The method presented in this thesis is a game-changer since it combines the accuracy of full annulus simulations with affordable computational costs. It aims at demonstrating that the proposed reduced-passage method can

provide solutions which are equivalent to full-annulus simulations with a significant reduction in computational resources as well as improving the methodology and understanding of unsteady and non-axisymmetric problems in different turbomachinery practical implementations.

The methodology is developed in the next chapter and the validation and discussion of the physical aspects are combined in the following chapters where the proposed method is applied and verified into several representative unsteady cases present in turbomachinery. This thesis is organised in following manner to cover the aforementioned objectives:

Chapter 1 presents an overview of the existing efficient computational methods to study unsteady and non-axisymmetric flows due to instabilities in turbomachinery and other related methods.

Chapter 2 describes the development and implementation of the proposed method to the resolution of linear wave equation, which will be used to assess the potential and limitations of the method. In addition, the practical implementations are generalized and implemented into a non-axisymmetric code.

Chapter 3 presents a engine representative non-axisymmetric configuration, such as a generic fan-stage, to explore the robustness and efficiency of the method as a validation case to analyse its behaviour under inlet distortion perturbations and stall conditions minimising the computational overhead of using full-annulus geometries.

Chapter 4 exploits the method to characterise the relevance of each nodal diameter in the fan stall process and study the non-linear stability in a harmonic by harmonic basis of the inlet total pressure and outlet static pressure perturbations.

Chapter 5 presents a validation of the proposed efficient method to study rotating instabilities in engine representative cavities. Non-axisymmetric and unsteady flows are approximated and analysed by the proposed method for three different cavity geometries.

Chapter 6 gives the conclusions of the thesis and some suggestions for further work.

2. Fundamentals of Passage-Spectral Method

Contents

2.1. One-dimensional Linear Wave Equation Model	24
2.2. Generalization to 3D Non-axisymmetric Geometries	50
2.3. Conclusions	59

This chapter presents the formulation of a novel efficient block-wise spectral reconstruction method. The approach is meant to be used in problems displaying a large disparity of spatial scales. In turbomachinery problems, the short scale is dictated by the airfoil pitch which gives rises to a fundamental harmonic with a wave-number associated with the blade count. This and its higher harmonics are resolved within the blade passage. This fundamental block is, in many occasions, modulated with a slowly varying spatial variation in the circumferential direction due to the airfoil interaction with upstream or downstream blade rows, or sometimes with long-wavelength spatial non-uniformities such as the engine intake.

Ideally, all the homologous points within the different blocks could be Fourier transformed to represent this spatial variation. This approach is not efficient since it would require performing as many Fourier transforms as number of points per passage per time step. Instead, since the baseline solver is designed in a pointwise manner, only the information on the circumferential boundaries of the block needs to be reconstructed to complete the numerical scheme.

It will be shown that the method can retain all the harmonics self-contained within the block

and the slow varying harmonics reconstructed using Fourier series only in the vicinity of the neighbouring blocks. Additionally, a third family of scattered or side-band spatial harmonics is captured by this approach. These modes are natively retained by the method.

In many practical problems, a few low-order spatial harmonics suffice to predict the solution accurately. This means that only a few blocks out of the tens or hundreds forming a full wheel of airfoils need to be solved with a remarkable reduction in the computational resources with respect to full-annulus simulations, giving the best compromise between sufficient accuracy of the results and the least computational effort. The efficient and accurate resolution of different spatial scales makes the method proposed in this chapter very well suited for several turbomachinery problems.

The numerical formulation of the method is introduced first using the one-dimensional wave equation. This model problem is used to help describing the harmonic content retained by the method since the spatial representation is a mixture of a finite volume method with a spectral reconstruction. Finally, the implementation of the so-called Passage-Spectral Method (PSM) in an unstructured edge-based finite-volume Reynolds-Averaged Navier-Stokes solver is described.

2.1. One-dimensional Linear Wave Equation Model

The one-dimensional Euler equations in conservative form can be written as

$$\frac{\partial}{\partial t} \begin{pmatrix} \rho \\ \rho u \\ \rho E \end{pmatrix} + \frac{\partial}{\partial x} \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho u H \end{pmatrix} = 0 \quad (2.1)$$

where ρ is the density, u the velocity, p the static pressure, $E = e + \frac{1}{2}u^2$ total energy, and $H = h + \frac{1}{2}u^2$ the total enthalpy. This system of equations has to be completed by an equation of state defining the thermodynamical properties of the fluid. Considering a perfect gas, $p = \rho R_g T$, $e = c_v T$ and $h = c_p T$. Equation 2.1 can be written in compact form as $\partial_t \mathbf{U} + \partial_x \mathbf{F}(\mathbf{U}) = 0$, where $\mathbf{U} = \{\rho, \rho u, \rho E\}^T$ is the vector of the conservative variables and $\mathbf{F}$ the flux.

2.1 One-dimensional Linear Wave Equation Model

Let consider the linearisation of the one-dimensional Euler equations about a uniform steady flow $\mathbf{U}_0$. In this case the flow can be expressed as $\mathbf{U} = \mathbf{U}_0 + \mathbf{u}(x, t)$ where $\mathbf{u}$ is a small perturbation ($\mathbf{u} \ll \mathbf{U}_0$). The linearised Euler equations, $\mathbf{u}_t + (\partial \mathbf{F} / \partial \mathbf{U})_0 \mathbf{u}_x = 0$, can be diagonalised and written in characteristic form as:

$$\frac{\partial}{\partial t} \begin{pmatrix} p + \rho_0 a_0 u \\ p - \rho_0 a_0 u \\ p - \rho a_0^2 \end{pmatrix} + \begin{bmatrix} u_0 + a_0 & 0 & 0 \\ 0 & u_0 - a_0 & 0 \\ 0 & 0 & u_0 \end{bmatrix} \frac{\partial}{\partial x} \begin{pmatrix} p + \rho_0 a_0 u \\ p - \rho_0 a_0 u \\ p - \rho a_0^2 \end{pmatrix} = 0 \quad (2.2)$$

where a_0 and u_0 are the sound speed and the convection velocity of the mean flow respectively. The resulting system of equations represents three independent linear wave equations for the characteristic variables, each of them with its own propagation velocity, $c_0^{(j)}$. The j -th characteristic equation can be written as $u_t^{(j)} + c_0^{(j)} u_x^{(j)} = 0$, and therefore the problem reduces to show how to solve the linear wave equation (LWE)

$$\frac{\partial u}{\partial t} + c_0 \frac{\partial u}{\partial x} = 0 \quad x \in (0, L), t > 0 \quad (2.3)$$

with a constant propagation speed, c_0 , subject to the initial condition

$$u(x, t = 0) = u_0(x) \quad (2.4)$$

and periodic boundary conditions at both ends of the computational domain

$$u(0, t) = u(2\pi, t) \quad (2.5)$$

The choice of these boundary conditions is based not only on its simplicity but on the fact that they are the natural boundary condition in the circumferential direction for a wheel of identical blades.

Figure 2.1.: Comparison of the PSM and the analytical solution ($k_L = 1$ and $k_c = N_B = 9$).

2.1.1. Motivation and Qualitative Description of the Method

The one-dimensional wave equation is an excellent vehicle to illustrate the whole purpose of the method due to its simplicity. The main idea of the method is sketched in Fig. 2.1 where the solution of the LWE has been initialised with a short wavelength, $\lambda_c = 2\pi/k_c$, disturbance superimposed with a long-wavelength, $\lambda_L = 2\pi/k_L$, perturbation having the latter a much longer wavelength than the former ($\lambda_L \gg \lambda_c$). The wave-number of the characteristic short-wavelength perturbation will be referred to as k_c . The spatial harmonics of both perturbations propagate at the same speed and do not interact between them since the problem is linear and non-dispersive. However, the longest spatial scale imposes a constraint in the minimum size of the computational domain. This problem appears in turbomachinery frequently.

The question that is addressed in this thesis is whether this problem can be tackled solving just a few small blocks of size λ_c and reconstructing the solution between them using Fourier series. Figure 2.1 shows in white regions blocks which are actually solved, and in grey the interpolated or reconstructed regions that are used just for displaying purposes. The black squares in the same figure represent the nodes which are solved.

The computational domain is subdivided into N_B identical blocks of size λ_c discretised with N_P mesh points each. The block size, which is equal to $2\pi/N_B$, is set to contain the short-wavelength λ_c and the higher spatial harmonics included within each block $k = nk_c$, where n is

an integer. On the other hand, the number of uniformly distributed blocks along with the whole domain, with spacing $s = 2\pi/(2M + 1)$, determines the number of spatial harmonics, M , of the block modulation reconstruction and enables the resolution of the long-wavelength harmonics with wave-number k_L .

Every single block is computed independently as a separate instance of the solver, and the blocks play the role of the sampling points in a Fourier transform. Different numerical schemes (e.g. finite difference, finite-volume, spectral methods, etc.) can be used to compute the solution inside each block but boundary conditions have to be provided at the interfaces with the neighbouring blocks to account for the long-wavelength variations. In this particular case the 1st spatial harmonic (see Fig. 2.1), with $k_L = 1$, and a short wavelength perturbation of the same size as the block ($k_c = N_B = 9$) are computed using only three blocks ($j = 1, 2$ and 3 white regions in Fig. 2.1). The existence of a disparity of length scales in the problem is explicitly exploited in the method to improve the computational efficiency.

The very same idea appears in several turbomachinery problems though. The short wavelength is always associated with airfoil perturbations and therefore has the size of the blade pitch. The generation of the long characteristic length is more subtle and is often associated to flow instabilities such as rotating stall or rotating disk instabilities, previously introduced in Chapter 1. The size of these perturbations is typically of the order of the rotor circumference, and the ratio between the long and short characteristic lengths of the order of the number of blades of the full wheel which oscillates between 20 and 150.

2.1.2. Method Harmonic Content

The purpose of this subsection is to foresee the harmonic content of the solution retained by the method. It is apparent that since not all the nodes or blocks of the full computational domain are solved the solution is necessarily filtered out. If each block is discretised with $N_P = 2N + 1$ points and $N_B = 2M + 1$ blocks the total number of harmonics is hence $N_P N_B / 2$.

Figure 2.2 shows the harmonic content resolution of the block Fourier transform in a schematic

Figure 2.2.: Schematic view of PSM harmonic content.

way, where the bars represent the harmonics that the method can capture and are within the resolution range of the method.

Four families of spatial harmonics are then included in the reconstruction of the full-domain solution using the Passage-Spectral Method:

1. The mean value or the 0-th harmonic.
2. $N = (N_P - 1)/2$ block embedded harmonics, $k = nk_c$, where n is an integer such that $n \leq N$. These inner modes are self-contained in the block domain and are not affected by any Fourier Transform. They are solved inside each block with the numerical method of choice. The fundamental short-wavelength perturbation is represented and named in Fig. 2.2 as k_c , while the higher-harmonics of the solution are represented by nk_c .
3. M external block modulating harmonics with wave-number $k = m$, where m is an integer such that $m \leq M$. These harmonics are reconstructed using $2M + 1$ uniformly spaced blocks to retain the fundamental long-wavelength harmonics k_L , shown as blue bars in Fig. 2.2.
4. $2MN$ scattered harmonics, $k = nk_c \pm m$, which are implicitly calculated by the method. These modes are shown in Fig. 2.2 as side-bands surrounding the nk_c harmonics.

The method was conceived originally to reproduce just the long- and short-wavelength harmonics, represented by the wave-numbers k_L and nk_c respectively. However, the block Fourier reconstruction approach followed by the Passage-Spectral Method has shown to be better than initially expected, and the scattered modes are natively retained in the method as well. This

fact turns the method into a powerful tool for turbomachinery applications where the scattered modes play a central role.

Turbomachinery configurations exhibit the so-called Tyler–Sofrin modes [68] or spatial harmonics $k = mN_b + nN_v$, where N_b represents the number of rotor blades and N_v refers to the total number of vanes. These are clearly visible in forced response or acoustic simulations steadily increasing the size of computational models consisting of multiple airfoil rows. The fundamental harmonic of the wake rotor with wave-number $k_L = N_b$, interacts with the vane perturbation, with $k_c = N_v$, to generate spinning-modes (or Tyler–Sofrin modes). These are nothing else than scattered modes. Depending on the actual blade count, these scattered modes can exhibit very long-wavelengths as well. What is relevant here is that the structure of the response of the physical and the numerical models are fully compatible.

2.1.3. Numerical Method Outline

The method is illustrated spatially discretising the LWE, $u_t + c_0 u_x = 0$, using a first-order finite-difference method because of its simplicity. However, the idea behind the method can be implemented in any solver with a small numerical stencil. The temporal discretization does not interfere with the method at all as long the scheme is explicit, or if implicit is based at least on marching in a pseudo-time, as it is the case of the dual time step method. The semi-discrete first-order form of the wave equation at the node i and block b can be written in the inner points of the mesh as:

$$\frac{\partial u_i^{(b)}}{\partial t} = -c_0 \left[\frac{u_{i+1}^{(b)} - u_{i-1}^{(b)}}{2\Delta x} + \mu \Delta x \left(\frac{u_{i+1}^{(b)} - 2u_i^{(b)} + u_{i-1}^{(b)}}{\Delta x^2} \right) \right] \quad (2.6)$$

where central differences have been used to discretise u_x , and numerical diffusion of the form $\mu c_0 \Delta x u_{xx}$ has been added to give a certain degree of upwinding character to the discretization. The baseline code is second-order accurate in space but follows a similar structure. For uniformly distributed mesh points, the mesh spacing is $\Delta x = x_i - x_{i-1}$, and μ is a dimensionless constant. Every block is computed in the solver separately, and the index i is local to the block.

Figure 2.3.: Sketch of the node and block index notation used in the one-dimensional numerical scheme.

Figure 2.3 introduces the nomenclature used in Eqn. 2.6 where $u_i^{(b)}$ represents the unknown in the i -th node of the block b . The index i is local to the block and nodes with identical i index are physically located at the same relative position within the block. These nodes are referred to as homologous nodes in the blocks. However, the block index b is global and spans from $b = 1, \dots, N_B$. As it has been mentioned before, the idea of the method is not to discretise all the blocks but just those corresponding to the indexes:

$$b = 1 + (j - 1)N_B / (2M + 1) \quad (2.7)$$

where j spans from $j = 1, \dots, 2M + 1$. Therefore, the block index j refers to the local and consecutive list of the blocks which are actually solved. The spatial discretization described in Eqn. 2.6 can be directly applied to all the inner points of the block. If the block is discretised using N_P points, the block length is $L = N_P \cdot \Delta x$. The node index i spans from $i = 1, \dots, N_P$ and hence the scheme in the block boundaries is:

$$\begin{aligned} i = 1 : \quad \frac{\partial u_1^{(b)}}{\partial t} &= -c_0 \left[\frac{u_2^{(b)} - u_0^{(b)}}{2\Delta x} + \mu \Delta x \left(\frac{u_2^{(b)} - 2u_1^{(b)} + u_0^{(b)}}{\Delta x^2} \right) \right] \\ i = N_P : \quad \frac{\partial u_{N_P}^{(b)}}{\partial t} &= -c_0 \left[\frac{u_{N_P+1}^{(b)} - u_{N_P-1}^{(b)}}{2\Delta x} + \mu \Delta x \left(\frac{u_{N_P+1}^{(b)} - 2u_{N_P}^{(b)} + u_{N_P-1}^{(b)}}{\Delta x^2} \right) \right] \end{aligned} \quad (2.8)$$

Figure 2.4.: Schematic of the block-wise spectral reconstruction of the 1st harmonic ($k_L = 1$) in a domain consisting of six blocks, $N_B = 6$.

where the nodes $u_0^{(b)}$ and $u_{N_P+1}^{(b)}$ do not form part of the block domain. If the block b were solved with periodic boundary conditions the problem would be closed by setting $u_0^{(b)} = u_{N_P-1}^{(b)}$ and $u_{N_P+1}^{(b)} = u_2^{(b)}$. In this case, the solution of the block is self-contained and the numerical method does not need to resort to out-of-block information.

In our case, the problem has not block-periodic boundary conditions, and therefore an alternative boundary condition has to be provided. If the information at all the nodes of the full domain were available these boundary conditions would be:

$$u_0^{(b)} = u_{N_P-1}^{(b-1)}, \quad u_{N_P+1}^{(b)} = u_2^{(b+1)} \quad (2.9)$$

However the nodes $u_{N_P-1}^{(b-1)}$ and $u_2^{(b+1)}$ belong to the neighbouring blocks (grey region in Fig. 2.3) and their value is unknown since these blocks are not contained in the discretization. In the present method the solution at the nodes $u_{N_P-1}^{(b-1)}$ and $u_2^{(b+1)}$ is reconstructed using the existing information in the $2M + 1$ existing blocks. Among the many different forms of reconstructing the signal in the full-domain, the most natural, and efficient in this case, is the use of discrete Fourier transforms. The Fourier coefficients of the inner homologous points at both periodic

boundaries are computed as:

$$\begin{aligned}\hat{u}_{Np-1}^m(t) &= \frac{1}{2M+1} \sum_{j=1}^{2M+1} u_{Np-1}^{(j)}(t) e^{-I 2\pi m j / (2M+1)} \\ \hat{u}_2^m(t) &= \frac{1}{2M+1} \sum_{j=1}^{2M+1} u_2^{(j)}(t) e^{-I 2\pi m j / (2M+1)}\end{aligned}\quad (2.10)$$

and used as interpolant functions to reconstruct the solution in the block boundaries. Figure 2.4 sketches the idea of the block-wise spectral reconstruction strategy. The Fourier coefficients in Eqn. 2.10 of the solution at the inner nodes of both periodic boundaries are calculated using the $u_{Np-1}^{(j)}$ and $u_2^{(j)}$ homologous boundary nodes in all the existing $2M+1$ blocks, (represented as filled red and blue squares in Fig. 2.4. The solution at the nodes $u_0^{(b)}$ and $u_{Np+1}^{(b)}$, located at the neighbouring blocks $b-1$ and $b+1$ respectively, are computed using the Fourier coefficients as:

$$\begin{aligned}u_0^{(b)} &= \sum_{m=1}^M \hat{u}_{Np-1}^m(t) e^{-I 2\pi m (b-1) / N_B} \\ u_{Np+1}^{(b)} &= \sum_{m=1}^M \hat{u}_2^m(t) e^{-I 2\pi m (b+1) / N_B}\end{aligned}\quad (2.11)$$

Taking this into account, the solution at the periodic boundaries of each block b can be expressed formally as a function of the inner solution at each sampling block j :

$$\begin{aligned}u_0^{(b)} &= f(u_{Np-1}^1, \dots, u_{Np-1}^{1+(j-1)K}, \dots, u_{Np-1}^{1+(2M)K}) \\ u_{Np+1}^{(b)} &= f(u_2^1, \dots, u_2^{1+(j-1)K}, \dots, u_2^{1+(2M)K})\end{aligned}\quad (2.12)$$

with spacing $K = N_B / (2M+1)$, where M is the number of harmonics in the Fourier transformation. The discrete Fourier transform used to reconstruct the solution in the block boundaries in Eqn. 2.12 is actually a linear combination of homologous points. Once the fluxes are computed and the variables in the blocks are updated, the solutions $u_{Np+1}^{(b)}$ and $u_0^{(b)}$ can be recomputed and the whole process repeated.

The local pointwise scheme within the uniformly spaced blocks to accommodate the in-block high-harmonic content with wavenumbers $k = nk_c$ enable the reconstruction of the solution without any loss of information. In other words, the harmonics contained within the blocks are

always well represented within the accuracy of the method employed to solve the blocks.

If M spatial harmonics are retained in the block Fourier transform, then just $2M + 1$ blocks out of a total number of N_B blocks need to be computed. Since only a neighbouring node at each periodic boundary of the j -th block is needed to construct the numerical scheme due to the compact stencil of the spatial discretization, only the Fourier coefficients of the adjacent inner nodes to the original periodic boundaries need to be evaluated. Since the number of points involved in the Fourier transform is proportional to the number of spatial harmonics, the method can eventually become inefficient if the number of retained harmonics is high. However, the fact that just a few harmonics are usually required in turbomachinery problems, and that the Fourier transforms need to be performed only in a small subset of mesh points makes the penalty due to the block-wise spectral reconstruction at the boundaries negligible.

The computational efficiency of the method is then proportional to the ratio between the total number of blocks and the number of the retained spatial harmonics in the simulations. The ideal speed-up factor, R , with respect the full-domain solution is then

$$R = \frac{N_B}{2M + 1} \quad (2.13)$$

The method is especially efficient when the number of blocks or blades passages is high. This number oscillates between 20 and 150 in realistic cases.

2.1.4. Spectral View of Block Reconstruction

The method can be more easily understood if it is regarded as a sequence of two aligned Fourier expansions in the same direction. Let considered that the governing equations within each block, were solved using a spectral method with periodic boundary conditions. In this case, the solution in the block b could be expanded in Fourier series as:

$$u^{(b)} = \sum_{k=-N_P/2}^{N_P/2} \hat{u}_k^{(b)} e^{I k x'} \quad (2.14)$$

where x' is the block local coordinate in the span $x' \in [0, 2\pi/N_B]$. If the solution is block periodic and all the blocks are identical, in the global coordinate system the solution can be expressed as

$$u = \sum_{k=-N_P/2}^{N_P/2} \hat{u}_k^{(b)} e^{I(kN_B)x} \quad (2.15)$$

If the solution is not block periodic and varies from block to block, the complex Fourier coefficients, $\hat{u}_k^{(b)}$, can be block-wise Fourier transformed to obtain:

$$u = \sum_{k=-N_P/2}^{N_P/2} \sum_{m=-M}^M \hat{u}_{k,m} e^{I(kN_B + m)x} \quad (2.16)$$

which is a double Fourier expansion in the same direction, x . This procedure is analogous to the one that would be conducted in a two-dimensional spectral solver with periodic boundary conditions where the variables are firstly Fourier transformed in the x -direction and then transformed again in the y -direction. Here the solution is transformed twice in the same direction.

In this context, the wave-numbers kN_B represent the block inner harmonics while the wave-numbers m account for the out-of-block harmonics, i.e. harmonics with long wavelengths which are not self-contained inside the block and modulate the block solution. There are $N_P/2$ inner harmonics and M outer harmonics. The simulation contains the spatial harmonics $kN_B + m$, which are a particular subset of all the potentially existent modes. Among all the existing harmonics, there are MN_P modes which are obtained as the combination of inner and outer wave-numbers. These are referred here as scattered modes since they are obtained physically from the interaction between the inner and outer scales.

It is important to highlight that since the first Fourier transformed is real but the second is complex $\hat{u}_{-k,m} = \hat{u}_{k,m}^*$ but $\hat{u}_{k,-m} \neq \hat{u}_{k,m}^*$. This means that both positive and negative m indexes contribute to the simulation and are independent.

In the present method, the “*inner*” Fourier transform does not exist since the method has not a spectral discretization. In practice, the inner block is solved using a second-order finite-difference or finite volume method (illustrated in this chapter using a 1st-order discretization

Figure 2.5.: Sketch of the spectrum content of the method for different values of the number of spatial harmonics, M , retained in the Fourier reconstruction.

for the sake of simplicity). However, the outer Fourier reconstruction is exact and can exactly replicate the inexact solution that would be obtained if the whole computational domain had been simulated.

Figures 2.5 (a), (b) and (c) illustrate the spectrum content as a function of the number of reconstructed harmonics, M , used by the Fourier transformation for $M = 1, 2$ and 3 respectively. Using $2M + 1$ uniformly spaced blocks M harmonics can be reconstructed. These harmonics are represented in the figure as k_L , close to the origin, and their number increases with M .

Embedded harmonics, which are self-contained inside each block, are noted as nk_c . Two of them, k_c and $2k_c$, are represented in Fig. 2.5. In addition, the scattered harmonics appear in

Figure 2.6.: Comparison of the PSM and full-domain solutions with an initial solution with $k_L = 2$. ($M = 2$, five computational blocks in white, and $N_B = 20$).

the spectrum as side-bands of the inner harmonics, $nk_c \pm m$, since the method is designed in such a way that $k_c \gg 1$. By increasing the number of outer harmonics the side-bands about the harmonics nk_c widen, as shown in Figs. 2.5 (b) and (c).

2.1.5. Numerical Results for the LWE

This subsection presents the application of the Passage-Spectral Method to the resolution of the one-dimensional wave equation with periodic boundary conditions. The objective is to validate and illustrate the conclusions outlined in the previous section. The ability of the scheme solving different initial value problems with different initial spatial harmonics, number of reconstructed spatial harmonics, M , and number of blocks N_B is displayed through examples.

The method solution is compared to the full-domain solution in a variety of cases to verify the results. Snapshots of solution are taken after several periods. The computational mesh consists of $N_P = 100$ points per block. Numerical parameters for the resolution of the LWE using the first-order scheme are $CFL = c_0 \Delta t / \Delta x = 0.1$ and $\mu = 0.5$.

The numerical scheme used to describe the method is simple and rudimentary but very high resolutions have been employed here to avoid contaminating the solution with issues related to the low quality of the underlying scheme which have nothing to do with the main idea of the method presented in this thesis.

Figure 2.7.: Comparison of the PSM ($M = 2$) and the full-domain solutions with and initial solution dcontaining two harmonics ($k_L = 2$ and $k_S = 15$). The total number of blocks is $N_B = 15$.

2.1.5.1. Single Outer Harmonic Case

First, a validation case is carried out to verify the correct reconstruction of a spatial harmonic k_L spanning outside of the block boundaries. Let consider the initial condition

$$u(x, t = 0) = \sin(k_L x) \quad (2.17)$$

with $k_L = 2$. The PSM method is compared to the full-domain solution (full annulus solution in the turbomachinery jargon) in the whole domain, being $N_B = 20$. The problem is solved using the PSM with $M = 2$, which means that five uniformly spaced computational blocks are used to calculate the 2^{nd} harmonic of the initial value problem in Eqn. 2.17.

Figure 2.6 represents the full-domain and the PSM solutions. The grey region displays the full-domain solution only while the white regions plot the solution of the PSM at each j -th block using the block-wise spectral reconstruction previously described. As shown in Fig. 2.6, both the full-domain and the PSM solutions are identical. The PSM reproduces the full-domain solution with the corresponding reduction in grid points and computational time since the method only solves 5 out of the $N_B = 20$ blocks to reconstruct the whole domain solution, which corresponds to a speed-up factor of 4.

2.1.5.2. Outer Harmonic Superimposed with Embedded Harmonics Case

Once it has been verified that the method can reconstruct solutions containing wavelengths longer than the block size a second perturbation with a wavelength contained within the block size is superimposed to the first. The initial conditions is:

$$u(x, t = 0) = \sin(k_L x) + \alpha \sin(k_S x) \quad (2.18)$$

being $\alpha = 0.4$, $k_L = 2$ and $k_S = 15$. The second term of the initial condition has been enforced to have a wavelength equal to the block size since the number of blocks in which the domain is subdivided is $N_B = 15$. The full-domain and the PSM solutions with two harmonics ($M = 2$) are compared in Fig. 2.7. Both solutions show the same sinusoidal variation as the imposed initial value with spatial variations inside the blocks as a consequence of the presence of the embedded harmonic k_c . The PSM solution can reproduce the two different spatial scales. The fundamental large-scale harmonic, k_L , is reproduced with the two spatial harmonics retained in the Fourier transform. The second related to the short-wavelength pattern and which is periodic inside each block mesh is solved by the inner block discretization since the harmonic k_c is spatially periodic in each block. This case corroborates the idea that the method can reproduce simultaneously wavelengths contained in the block and wavelengths longer than the block size.

The aim now is to verify that the proposed PSM can reproduce any number of higher periodic harmonics inside each block domain as long as they are supported by the internal discretization of the block. The LWE is solved with the following initial condition:

$$u(x, t = 0) = u_0(x) = \alpha \sin(k_L x) + \sum_{b=1}^{N_B} e^{-a[x - (2b-1)\frac{\pi}{N_B}]^2} \quad (2.19)$$

which represents a sinusoidal wave modulating a solution which is compact within the blocks (second term in Eqn. 2.19) if the parameters are properly chosen. The amplitude of the 1st spatial harmonic ($k_L = 1$) has been set to $\alpha = 0.2$. The second term in Eqn. 2.19 represent a summation of a compact Gaussian function if a is large enough. The value of a has been set to

Figure 2.8.: Comparison of PSM ($M = 1$, three computational blocks) and full-domain solutions (a) and harmonic Fourier content (b) with an initial solution described in Eqn. 2.19, where $k_L = 1$, $N_B = 9$, $\alpha = 0.2$ and $a = 10$.

$a = 10$ to ensure that the tails of the Gaussian do not extend beyond the limits of the blocks.

It could be argued that this case is irrelevant since if the problem is linear the results could be presented in a harmonic-by-harmonic basis. However, it was thought that displaying results with a more “physical” initial condition could help to grasp the engineering applications of the method.

Figure 2.8 shows the comparison between the PSM ($M = 1$) and full-domain solutions in a case where $N_B = 9$. The white regions in Fig. 2.8 (a) represent the computational domain for the PSM. In these zones, the solution of the PSM is compared to that of the full-domain solution. Both solutions are shown to be identical. Figure 2.8 (b) shows the harmonic content of both PSM and full-domain solutions. The block-to-block deviation introduced by the 1st spatial harmonic of the initial solution is clearly seen in Fig. 2.8 (a). Figure 2.8 (b) displays the harmonic content of the initial solution and the solution after several wave passings. The initial condition Eqn. 2.19 contains high-order harmonics embedded inside each block. It is interesting to verify if the method can capture all these block-embedded harmonics nk_c . These correspond in Fig. 2.8 (b) with the Fourier indexes 9, 18, 27, etc. It is concluded that the block-wise spectral reconstruction reproduces all the block-embedded spatial harmonics nk_c for any integer $n \leq (N_P - 1)/2$ inherently.

It is apparent that the accuracy of the high-order harmonics will be low, especially if a 1st-order method is used. However, the PSM is transparent in this regard, and the accuracy of the original method implemented in the blocks is preserved.

2.1.5.3. Non-Matching Block Case

Since the number and size of the existing physical blocks are given, the use of uniformly spaced discrete Fourier transforms leads to the use of blocks which are not necessarily fitting with none of the existing blocks. This fact may contribute to increasing the error with respect the cases presented up to now where the number of blocks was chosen in such a way that the blocks used to reconstruct the signal correspond to “physical” blocks. This degree of freedom does not exist in practice where the number of blocks is given by the number of blades.

Figure 2.9.: Comparison of PSM ($M = 1$, three computational blocks) and full-domain solutions (a), PSM-Reconstructed solution (b) and harmonic Fourier content (c) with the initial solution described in Eqn. 2.19, where $k_L = 1$, $N_B = 10$, $\alpha = 0.2$ and $a = 10$.

To shed some light about the case when the physical position of some of the $2M + 1$ block samples in the PSM does not coincide with the real position of the N_B blocks in the full-domain, a case with three block samples ($M = 1$) to retain the 1st harmonic, and a total number of blocks $N_B = 10$ is studied. In this case, the collocation of harmonic $k_L = 1$ with a single mode ($M = 1$) requires the use “unphysical” non-fitting blocks with the original mesh. Figure 2.9 (a) shows the PSM and full-domain solutions for this case. It can be seen that in the PSM the physical positions of the second ($j = 2$) and third ($j = 3$) blocks do not correspond with any of the real blocks of the full computational domain. The position of these virtual blocks is imposed by the need of using an equispaced Fourier transform with a minimum number of blocks (three in this case). The solution of the PSM appears displaced with respect one of the full-domain solution since the blocks are in a different position. Nevertheless, the local amplitude of the 1st harmonic is retained. However, when the solution is properly reconstructed and compared to the full-domain solution, the matching is excellent.

Once the solution in the uniformly spaced blocks is calculated, the full-domain solution is reconstructed Fourier transforming all the homologous nodes of the blocks. In the example presented here, only the 1st harmonic can be obtained. The discrete Fourier series of each mesh node is then used as an interpolant function to derive the solution in the physical nodes. This Fourier series can be seen as an expansion of the first virtual block ($j = 1$) which is always chosen to match the first physical block.

Figure 2.9 (b) shows the comparison of the reconstructed PSM solution to the full-domain solution using the three uniformly spaced blocks to reconstruct the complete solution. Despite that the blocks are not placed in the same physical position, both solutions are identical, as shown in Fig. 2.9 (b) in the physical plane, and Fig. 2.9 (c) in the Fourier domain. It is concluded the PSM can calculate the domain-embedded harmonics even if the virtual blocks are not matching the physical blocks.

Figure 2.10.: Comparison of PSM ($M = 2$, five computational blocks) and full-domain solutions with an initial solution described in Eqn. 2.18 where $k_L = 2$, $k_S = 13$, $N_B = 15$ and $\alpha = 0.4$.

2.1.5.4. "Scattered" Mode Case

A case to validate the resolution of the previously outlined scattered or side-band harmonics $nk_c \pm m$ is included here. The aim is to verify the accuracy of the block-wise spectral reconstruction of mixed modes obtained by a combination of Fourier reconstructed and block resolved parts of the solution. These modes correspond to the sidebands sketched in Fig. 2.2.

In this particular case, the initial condition consists of two sines with different wavelengths, as in the previous case (see Eqn. 2.18), being $\alpha = 0.4$ and $k_L = 2$, where the domain is subdivided into $N_B = 15$ blocks. However, in this case, the second term of the initial value in Eqn. 2.18 corresponds to the spatial harmonic $k_S = 13$. This mode is neither directly included in the Fourier reconstructions nor fitting inside a single block. Therefore, the ability of the method to deal with this type of problem deserves to be pursued.

The full-domain and PSM solutions with two harmonics ($M = 2$) are compared in Fig. 2.10. It can be appreciated that both solutions are identical. This is consistent with the basic idea that the method can deal not only with the block-embedded modes, in this case, $k_c = N_B = 15$ and all its multiples but also with their side-bands. In this case since $M = 2$, the mode $k_S = k_c - m$, with $m = 2$, is retained in the solution. The modes 13, 14, 15, 16 and 17 would be reproduced accurately as well if they were present in the solution. This case validates the idea that the method can reproduce as well scattered or side-band harmonics.

Figure 2.11.: Comparison of the PSM ($M = 2$) and the full-domain solutions with the initial solution described in Eqn. 2.18 where $k_L = 2$, $k_S = 5$, $N_B = k_c = 15$ and $\alpha = 0.4$.

2.1.5.5. Out-of-spectrum modes

The limitations of the proposed method are explored in the following lines. Let simulate now a initial value problem consisting of two different sinusoidal perturbations with different wave-lengths again. The form of the perturbation is exactly the same as the one described in Eqn. 2.18 being in this case $\alpha = 0.4$ and $k_L = 2$. The domain is subdivided into $N_B = 15$ blocks. The wavenumber of the second term in Eqn. 2.18 is set to $k_S = 5$. Figure 2.11 shows that the PSM solution with two harmonics ($M = 2$) is quantitatively different than the full-domain solution. The PSM cannot reproduce the mode $k = 5$ since it is not included in any of the three family modes handled by the method.

Figure 2.11 (b) shows the harmonic content of the PSM solution. The mode $k = 5$ which is not contained the method's spectrum, for this particular setting of the parameters, is aliased in other low- and high-order harmonics. It is concluded that the solution of the PSM has little do with that of the full-domain.

2.1.6. Solution of the Wave Equation with varying propagation speed

The homogeneous linear wave equation has not any built-in characteristic length apart from the domain size. Somehow, the method has been artificially illustrated, setting different initial conditions providing different characteristic lengths. Block sizes have been arbitrary set to illustrate the method since there is not a natural way to chose them. This is not the case in turbomachinery problems where the block size is determined by the blade pitch, and this is no longer a degree of freedom.

The wave equation with variable propagation speed $c(x)$

$$\frac{\partial u}{\partial t} + c(x) \frac{\partial u}{\partial x} = 0, \quad (2.20)$$

is an interesting vehicle to test the method since it enables the possibility of including an intrin-

Figure 2.12.: Analytical solution (a) in Eqn. 2.23 and harmonic Fourier content (b) with $k_L = 1$, $k_c = 6$ and $\beta = 0.5$.

sic characteristic length in the spatial variation of the propagation speed. Moreover, Eqn. 2.20 gives rise to scattering, i.e. pure sinusoidal waves are not a solution to the problem anymore. Incoming or existing waves in the domain are dispersed by $c(x)$ creating waves with wavelengths different from the original one. The variable speed $c(x)$ plays the role of a non-uniform field created supposedly by a row of blades.

Among the many potential choices for $c(x)$ in this section the following expression has been chosen:

$$c(x) = \frac{c_0}{1 + \varepsilon \cos(k_c x)} \quad (2.21)$$

where ε controls the level of the non-uniformity and $\lambda_c = L/k_c$ is the wavelength of the variation of the propagation velocity. The reason for the selection of this form of $c(x)$ is threefold. Firstly, eases the analytical solution of the problem, secondly, the variation is spatially periodic and thirdly, enables the analytical determination of the Fourier content of the solution.

Finally, the problem is solved with the following initial condition:

$$u(x, t = 0) = \sin(k_L x) \quad (2.22)$$

The solution of Eqn. 2.20 with the propagation speed given by Eqn. 2.21 is readily obtained if the following change of variables is performed $d\eta = (1 + \varepsilon \cos k_c x)dx$ to reduce the problem to that of solving the LWE with constant velocity in the transformed space, i.e. $u_t + c_0 u_\eta = 0$.

The solution can be obtained then substituting $\eta = x + \frac{\varepsilon}{k_c} \sin(k_c x)$ in the solution of the transformed wave equation. The exact solution of the wave equation with the propagation velocity defined in Eqn. 2.21 and the initial condition in Eqn. 2.22 after an initial transient is:

$$u(x, t) = \sin[k_L(x - c_0 t) + \beta \sin(k_c x)]. \quad (2.23)$$

being $\beta = \varepsilon \frac{k_L}{k_c}$. This particular solution of the varying speed wave equation is triggered by the initial condition and obtained, as it has been mentioned before, after a transient which is not described here. This analytical solution is used to compare the results obtained using the PSM

and full-domain computation. The harmonic content of the solution is derived in Appendix A, where it can be seen how the modulation of the harmonic k_L in Eqn. 2.23 gives rise to sidebands around the harmonic k_c , as shown in Eqn. A.7 and Fig. A.1 in Appendix A for different values of the modulation index β .

Figure 2.12 shows the solution of Eqn. 2.23 for $k_L = 1$, $k_c = 6$ and $\beta = 0.5$. The solution can be expressed as a function of Bessel functions of the first-kind $J_n(\beta)$ (see Eqn. A.7 in Appendix A). The modulation of the mean flow velocity $c(x)$ in Eqn. 2.21 gives rise to the formation of scattered harmonics. It is important to highlight that the total energy by summation of each harmonic contribution is constant, as can be calculated from the harmonic components in Fig. 2.12 (b) and equal to 1. This is not surprising, since we have not included damping or dissipative forces. This can be also demonstrated mathematically by the properties of the Bessel functions of first kind described in Eqn. (A.8) in Appendix A.

The main objective of this section is to elucidate if the method can accurately capture these scattered harmonics as it has been previously claimed. The PSM solution with one spatial harmonic (three blocks) is now compared against the full-domain and analytical solutions. In this case, $k_c = N_B$ to force the blocks contain the periodic spatial variation of the propagation speed. Figure 2.13 shows a comparison between PSM, full-domain and analytical solutions for $M = 1$ and two different values of the harmonic of the spatial perturbation, k_c , with a fixed value of $\varepsilon = 0.8$ (see Eqn. 2.21). Figures 2.13 (a) and (b) display the comparison of the three solutions for $k_c = 4$ and 6, respectively. The good agreement between them allow us to conclude that the PSM can capture the interaction between the non-uniform mean flow velocity with spatial harmonic k_c , and the imposed harmonic k_L in the initial value in Eqn. 2.22.

Figures 2.13 (c) and (d) show the harmonic content of the three solutions displayed in Figs. 2.13 (a) and (b), for $k_c = 4$ and 6 respectively. Both cases exhibit sidebands of the modulating harmonic k_c , and the spectra contain a carrier harmonic of amplitude close to 1 for the wavenumber $k = 1$, due to its presence in the initial solution. These results can be directly compared with the ones obtained in Appendix A for the corresponding values of the parameter $\beta = \varepsilon \frac{k_L}{k_c}$. In the cases studied in this work, the modulation index is very small, $\beta \ll 1$, since we are targeting

Figure 2.13.: Comparison of the PSM ($M = 1$, three computational blocks), the full-domain and the analytical solutions (sub-figures (a) and (b)) and their harmonic contents (sub-figures (c) and (d)) for the case $\varepsilon = 0.8$ and $k_L = 1$, and two different wavenumbers of the propagation speed, $k_c = 4$ and $k_c = 6$. The wavelengths of the propagation speed and the block size are identical, $N_B = k_c$.

the limit $k_L \ll k_c$. and then the amplitude of the carrier harmonic k_L , which is $J_0(\beta)$ is approximately the unity. On the other hand, since we have a modulated wave, two more harmonics appear around the modulated harmonic k_c , in the form of sidebands, as shown in Figs. 2.13 (c) and (d) for $k_c = 4$ and 6, respectively. These sidebands in the spectra correspond to the scattered harmonics $k_s = k_c \pm m$, which according to the conclusions derived in the Appendix A for $\beta \ll 1$ correspond with the first Bessel function $J_1(\beta) \simeq \frac{\beta}{2}$. In principle, an infinite number of harmonics is produced by the interaction of the initial condition and the non-uniform mean flow perturbation $c(x)$. However, since the modulation index is small, higher-order side-bands, $k_s = nk_c \pm m$, are very small because their amplitude decay faster than $\mathcal{O}(\beta^n)$ if $\beta \ll 1$. This is seen for instance in Fig. 2.13 (c) for $k_c = 4$, where the harmonics 7 and 9 are hardly identified in the spectrum. Although the amplitude of the side-bands decays very quickly, the PSM can retain all the scattered harmonics. This conclusion is not limited to cases with $\beta \ll 1$ and holds as well when $\beta \sim \mathcal{O}(1)$ though this case has not been illustrated here.

2.2. Generalization to 3D Non-axisymmetric Geometries

Once the method has been verified in a one-dimensional problem, the methodology is implemented in a finite-volume compressible URANS flow solver [69, 70, 71, 72, 73, 74, 75]. First, a brief introduction to the main features of the existing solver is provided. The basic modelling assumptions and implementation aspects for the spatial Fourier modelling approach in a generic non-axisymmetric code are presented and discussed next. The developments presented in this thesis, especially those related with spatial Fourier method and its implementation, are based on the previously existing solver known as Mu^2s^2T , and that has been the subject of several PhD thesis and an intense development by the ITP Aero Simulation Department, formerly the Technology and Methods department.

Figure 2.14.: Hybrid cell grid and associated dual mesh.

2.2.1. Baseline Flow Solver

The three-dimensional compressible Navier-Stokes equations in conservative form for an arbitrary control volume can be written in compact form as:

$$\frac{d}{dt} \int_{\Omega} \mathbf{U} d\Omega + \int_{\Sigma} \mathbf{F}(\mathbf{U}) \cdot d\mathbf{A} = 0 \quad (2.24)$$

where $\mathbf{U}$ is the vector of conservative variables, $\mathbf{F}$ the sum of the inviscid and viscous fluxes, Ω the volume of the control volume, Σ its boundary and $d\mathbf{A}$ the differential area pointing outward to the boundary.

The baseline solver, known as *Mu²s²T* (Multirow Unsteady Unstructured Specific Solver for Turbomachinery) [76], discretises the spatial domain using hybrid unstructured grids and can contain cells with an arbitrary number of faces. The governing Eqn. 2.24 are discretised applying a finite volume approach based on the dual mesh where the solution vector stores the variables at the vertexes of the mesh. The control volume associated with a node is formed by connecting the median dual of the cells surrounding it, using an edge-based data structure. The spatial discretization is formally second-order accurate, and a matricial form of the numerical dissipation terms is used recurring to the Roe's matrix. The code is parallelised using MPI and executed in GPUs (GeForce GTX 1080 Ti) to reduce the turn around time [77].

The discrete equations are obtained from Eqn. 2.24 using a finite volume formulation, where the control volumes are constructed making use of the dual mesh (see Fig. 2.14), that is, the

mesh that results from connecting the centroids of the cells that surround a node [70, 78]. Thus, the solution is stored in the cell vertex, and the fluxes over the control volume faces are obtained computing the semi-sum of the fluxes in each node, which yields a second-order cell-centered scheme in the dual mesh.

Once the spatial domain is discretised, the semi-discrete version of non-linear Eqn. 2.24 for the internal node i can be written as:

$$\Omega_i \frac{d\mathbf{U}_i}{dt} + \sum_{j=1}^{n_{edges}} \left(\mathbf{F}_{ij}^{I+V} \cdot \boldsymbol{\sigma}_{ij} - \mathbf{D}_{ij} \right) = \mathbf{S}(\mathbf{U}_i) \quad (2.25)$$

where Ω_i is the control volume, $\boldsymbol{\sigma}_{ij}$ is the area associated with the edge ij , $\mathbf{F}_{ij}^{I+V}$ represents the inviscid and viscous fluxes through this area, $\mathbf{D}_{ij}$ are the artificial dissipation fluxes, $\mathbf{S}$, represents the source terms, and n_{edges} the number of edges surrounding the node i . Turbulence effects are accounted for using the $k - \omega$ model with realizability. An enhance wall function model for the boundary layer is also used.

Equation 2.25 can be written in compact form as:

$$\frac{d\mathbf{U}_i}{dt} = \mathbf{R}(\mathbf{U}) \quad (2.26)$$

Several techniques are available to accelerate the convergence to the steady state, $\mathbf{R}(\mathbf{U}) = 0$. However, in unsteady problems the system of Eqn. 2.26 needs to be solved in a time-accurate manner. Explicit time marching algorithms have sometimes very stringent stability restrictions and in these cases implicit methods are preferred to solve the system. This situation is presented when the time scales we are interested in are much longer than the shortest time scale that is solved in the simulation. This is typically the case when the stability restrictions that arise due to the spatial accuracy requirements impose a time step much shorter than the one that would be selected based solely on temporal accuracy requirements.

A second-order backward implicit time integration of Eqn. 2.26 can be expressed as:

$$\frac{3\mathbf{U}_i^{n+1} - 4\mathbf{U}_i^n + \mathbf{U}_i^{n-1}}{2\Delta t} = \mathbf{R}(\mathbf{U}^{n+1}) \quad (2.27)$$

where n is the time level. The resulting system of non-linear equations is solved using a dual-time step, τ , adding the pseudo-derivative $\mathbf{U}_\tau$ [79] to Eqn. 2.27, and preconditioning the system to speed up the convergence to the pseudo-steady state $\mathbf{U}_i^{n+1}$

$$P_{ji}^{-1} \frac{d\mathbf{U}_i}{d\tau} = \mathbf{R}^*(\mathbf{U}) = \mathbf{R}(\mathbf{U}) - \frac{3\mathbf{U}_i - 4\mathbf{U}_i^n + \mathbf{U}_i^{n-1}}{2\Delta t}. \quad (2.28)$$

The superscript n refers to the physical time step, thus when the steady state of the problem is reached, what we have actually obtained is the solution of the unsteady problem in $n + 1$. The discretization written in Eqn. 2.28 is implicit when marching in the physical time but explicit when marching in the dual one. When this new problem is converged to a steady state, Eqn. 2.26 is recovered. Thus, the problem has been reduced to achieve the solution of a steady problem with an additional source term.

Equation 2.28 can be solved using steady state acceleration techniques, marching in τ instead of t . The preconditioning matrix P_j^{-1} can include block-Jacobi and/or low Mach number preconditioning, multi-grid, and any other mechanism to speed-up the convergence. The acceleration mechanisms used to remove the time-step limitations of the explicit scheme can be regarded as an inner iteration of the method, and, in principle, are transparent to the method propose here.

The vectors of unknowns, $\mathbf{U}$, and residuals, $\mathbf{R}$, in a full annulus simulation can be split in a block basis as

$$\mathbf{U} = \begin{pmatrix} \mathbf{U}_i^1 \\ \mathbf{U}_i^2 \\ \cdot \\ \cdot \\ \cdot \\ \mathbf{U}_i^{N_B} \end{pmatrix}, \mathbf{R} = \begin{pmatrix} \mathbf{R}_i^1 \\ \mathbf{R}_i^2 \\ \cdot \\ \cdot \\ \cdot \\ \mathbf{R}_i^{N_B} \end{pmatrix} \quad (2.29)$$

where the column vector $\mathbf{U}^{(j)}$, contains just the unknowns of the j -th blocka and N_B is the number of blocks. The node i represents homologous nodes in different blocks or passages. In a generic non-axisymmetric case in turbomachinery, the number of rotor blades or vanes of the row define a block-passage. In the following lines, we refer to a reduced-passage sample to note

a non-axisymmetric block.

2.2.2. 3D Block-wise Spectral Method

The present section is aimed at extending the Fourier modelling to 3D non-axisymmetric geometries and unsteady flows subject to circumferential distortions. The homologous nodes, of the different passages, $\mathbf{U}_i^{(j)}$, can be regarded as a discrete series of points which can be expressed in terms of its Fourier coefficients as

$$\hat{\mathbf{U}}_i^m(\mathbf{x}, t) = \frac{1}{2M+1} \sum_{j=1}^{2M+1} \mathbf{U}_i^{(j)}(x, r, \theta_j, t) e^{-Im\theta_j} \quad (2.30)$$

whereby the origin of the j -th passage is located at θ_j (previously defined in Chapter 1):

$$\theta_j = \frac{j-1}{2M+1} 2\pi \quad (2.31)$$

Here, the methodology is applied to periodic perturbation in the azimuthal direction, θ , following the same logic as in the one-dimensional problem for the periodic waves in the axial direction. If all the passages of the wheel are used then $(N_B - 1)/2$ different complex Fourier coefficients, or harmonics, $\hat{\mathbf{U}}_i^m(\mathbf{x}, t)$, are obtained at each node i , and the approximation of the whole wheel or annulus is exact.

Fourier series of infinitely continuous and differentiable functions converge very quickly (exponentially), and therefore, very accurate representations of the solution can be obtained with just a few harmonics. The purpose of using Fourier modelling of the spatial flow-field is to use a truncated Fourier series of the “exact” solution and retain a reduced number of spatial harmonics, M .

The underlying hypothesis is that the wavelength of the circumferential non-uniformities is long, and therefore, a few harmonics suffice to represent the flow-field accurately. If only M harmonics are retained then just $2M + 1$ passages need to be computed instead of N_B . The idea of the method is sketched in Fig. 2.15 where it can be seen that the 1st harmonic of the

Figure 2.15.: Schematic of a 1ND sinusoidal distribution in whole annulus (left) and its approximation (right) with three reduced-passages samples (dark grey) uniformly spaced and not in the physical position of the original full-annulus.

circumferential variations of the flow-field is reconstructed computing only three passages (in dark grey). The solution in the rest of the passages (in light grey) is interpolated using a Fourier reconstruction containing solely the information of a few single-passage domains. The method is aimed at reproducing long-wavelength patterns, of the size of the whole circumference, and short wavelength disturbance, of the order of the pitch, which are absorbed within the block-passage mesh, as shown in Fig. 2.15 by dark grey passages. The long-wavelength is then represented using the Fourier series sampled by the uniformly spaced block-passages along the circumference.

Equation 2.25 can be re-written taking into account the nomenclature introduced in Eqn. 2.29 as:

$$\frac{d}{dt} \begin{pmatrix} \Omega_i \mathbf{U}_i^1 \\ \Omega_i \mathbf{U}_i^2 \\ \cdot \\ \cdot \\ \cdot \\ \Omega_i \mathbf{U}_i^{2M+1} \end{pmatrix} + \begin{pmatrix} \mathbf{R}_i^1 \\ \mathbf{R}_i^2 \\ \cdot \\ \cdot \\ \cdot \\ \mathbf{R}_i^{2M+1} \end{pmatrix} = \begin{pmatrix} \mathbf{S}_i^1 \\ \mathbf{S}_i^2 \\ \cdot \\ \cdot \\ \cdot \\ \mathbf{S}_i^{2M+1} \end{pmatrix} \quad (2.32)$$

where the residual in the node i , and passage j , is:

$$\mathbf{R}_i^j = \sum_{j=1}^{n_{edges}} \left(\mathbf{F}_{ij}^{I+V} \cdot \boldsymbol{\sigma}_{ij} - \mathbf{D}_{ij} \right)^j$$

It is important to remark that while the source term at node i , $\mathbf{S}_i$, has point-wise description, the residual $\mathbf{R}_i$, involves several layers of neighbouring nodes to compute the fluxes.

Figure 2.16.: Updating periodic ghost cells.

First, Fourier coefficients are calculated in the inner nodes for both periodic boundaries, as it is shown in Fig. 2.16 for a schematic of a two-dimensional case. The evaluation of the flow variables at the two neighbour rows of mesh cells is required because of the numerical stencil of structured codes with 4th order artificial viscosity terms involves at least the neighbours of the neighbours in the computation of node i , when switchers and limiters are included, the stencil of the spatial discretization is even larger, and more neighbouring nodes are involved. Moreover, all explicit time-marching methods, and most of the implicit ones, are local and do not require the use of nodes located far away of the node of interest. This is especially true in codes designed to run in parallel architectures efficiently, where the communication among neighbouring partitions is minimised by design.

A phantom mesh at the periodic patches is used in order to evaluate new flow variables with the corresponding phase-shift Fourier transformation by duplicating points in the vicinity of the periodic boundaries on the opposite boundary. Figure 2.16 shows the control volume associated with a node i , located in the passage periodic boundary. The control volume is formed by connecting the median dual of the cells which surround it. It is seen that the computation of the residual in the nodes located in the vicinity of the periodic boundary involves nodes of other

passages or blocks.

The method requires the reconstruction of the solution in the vicinity of the nominally periodic boundaries with the neighbouring blocks $b + 1$ and $b - 1$, as previously shown for the one-dimensional problem in Fig. 2.4. Figure 2.16 displays the halo of cells which needs to be interpolated using the information of all the computed passages or blocks using the Fourier reconstruction with the location of the new cells taking into account the phase-shift, which is equal to the pitch P of each block b . The reason to use the pitch of the sample in the interpolation of the new values at the ghost mesh using the Fourier coefficients calculated at the opposite periodic boundary, is the non-axisymmetric nature of the mesh. Once the conservative variables have been updated, the corresponding correction to the gradients and fluxes at each time-step iteration is added. Since the numerical scheme is local in nature, just a small fraction of the total number of the passage nodes needs to be Fourier transformed and, therefore, the computational penalty in the global algorithm is minimal.

A similar approach was followed by He to compute the non-axisymmetric flow and heat transfer in a rotating cavity [62], but in this case, the fundamental flow unit was a two-dimensional slice of the computational domain. He also used the same idea to retain long-wavelength spatial circumferential variations due to intake distortion or fan-stator/OGV interactions using a spatial Fourier expansion [61]. However, the application of this idea to compute 3D flow instabilities, instead to flows with forced or prescribed circumferential variations, either due to the geometry or the boundary conditions is new. He [62, 61] developed a proof of concept using structured meshes, in this research fully unstructured grids are used for the sake of generality, what makes the practical implementation more complex.

2.2.3. Practical Implementation

Every passage or block is computed as a separate instance of the baseline code. However, the periodic boundary condition needs to be generalised correcting the computation of the fluxes to account for the neighbouring passages. At any time level, n , the Fourier series of the flow variables of the halo or the ghost mesh, as shown in Fig. 2.16 for a one-dimensional problem,

is computed using only M spatial harmonics or $2M + 1$ homologous nodes. This Fourier series is then used as an interpolant function to compute all the node i

$$\mathbf{U}_i^{(b)}(\mathbf{x}, t) = \sum_{m=1}^M \hat{\mathbf{U}}_i^m(\mathbf{x}, t) e^{Im\theta_{i,b}} \quad (2.33)$$

where $\theta_{i,b}$ represents the azimuthal position of the node i of the ghost node in the b -th passage. Every block, or passage, has a halo of cells representing the missing points of the neighbouring passages. Once the variables in the halo mesh have been reconstructed, the fluxes in the boundaries are computed following the same scheme as the inner nodes (see Fig. 2.16)

This methodology can be easily implemented in a general purpose code, since we only need to substitute the periodic correction of the fluxes and the gradients by the Fourier reconstructed values of the conservative variables in the neighbouring blocks, with a minimum modification of the flux calculation routine. This kind of methodology is applicable to the simulation of compressor instabilities (rotating stall) and the long-wavelength instabilities which can be seen in the disk drum and under-platform cavities of turbines and compressors. This approach will be verified against full-annulus cases in the following chapters for different applications in turbomachinery, with a remarkable reduction in computational resources.

The computational efficiency of the method is directly related to the ratio between the number of the retained spatial harmonics in the simulations, M , and the number of blades or reduced-passages, N_B . The ideal speed-up factor, R , with respect the direct full annulus solution was previously defined in Eqn. 2.13. The domain size will vary with blade count ratios and wave numbers but speed-up factor diminish with the number of spatial harmonics needed to approximate the circumferential disturbances.

Post-processing Once the solution of the three uniformly spaced reduced-passages is calculated at each time step (see Fig. 2.15), the reconstructed full-annulus solution can be obtained Fourier decomposing the flow variables at all the nodes of the sample passages. Once the Fourier coefficients at all the nodes of the passage are obtained these can be used to expand the solution to the full circumference. Since the reduced-passage samples play the role of sampling

points for the circumferential Fourier transform and require uniform spacing, the circumferential location of these sample passages does not correspond in general with the position in the full-annulus of the physical blades or passages. The Fourier coefficients use as reference the first reduced-passage ($j = 1$ in Fig. 2.15), which has the same physical location as the original full-annulus passage. The solution at the rest of the blades/passages of the full circumference is reconstructed using the Fourier coefficients. This step is only needed for displaying purposed or post-processing.

2.3. Conclusions

A block-wise spatial Fourier transform for the modelling of three-dimensional non-axisymmetric flows is proposed for time-domain unsteady flow solutions. The practical implementation of the proposed block Fourier transform has been first described using the one-dimensional wave equation and described in detail. This novel strategy brings the opportunity to calculate accurately different spatial harmonics just solving a fraction of the full-domain domain, which is subdivided into blocks, with the corresponding savings in computational time and mesh size. This reconstruction of the full-domain domain has the same level of accuracy as the original full-annulus approach is exact if enough spatial harmonics are retained.

It has been shown that the novel block-wise spectral reconstruction approach, can accurately capture three different families of spatial harmonics namely, the fundamental harmonics explicitly retained in the Fourier transform, the embedded harmonics self-contained in the blocks, and the scattered harmonics which are natively retained in the method as well. This means that the proposed block-wise spectral reconstruction strategy is fully compatible with the structure of the Tyler-Sofrin modes, converting the method into a powerful tool for other applications.

The method has been applied to the one-dimensional wave equation with periodic boundary conditions and compared with the full-domain domain solutions. The agreement between both solutions is high if the number of harmonics is high enough.

It is important to keep in mind, which is the harmonic content that the method can retain before

applying it to solve a particular problem. This raises the question of when and how using the proposed method. If this information is used properly, and only the harmonics of interest are calculated, the method is very efficient since only a few computing blocks are needed.

The formulation has been finally implemented in an existing implicit/explicit finite volume Navier-Stokes code, and its generalization for non-axisymmetric geometries is described in detail. As the flow non-uniformities in the circumferential direction have long length scales, only a small number of circumferential Fourier harmonics is needed, resulting in a drastic reduction in the number of mesh points required for such analyses. The solution of the reduced-passages samples can be Fourier transformed, and the Fourier coefficients used to build the solution in the full circumference. The method does not require any hypothesis about the temporal periodicity of the flow and can be easily implemented in time-marching CFD solvers with minimum modifications on the flux calculation routine since the information needs to be reconstructed just in the boundaries with the neighbouring blocks.

3. Unsteady Flows in a Fan-Stage

Contents

3.1. Forcing Due to Inlet Distortion	62
3.2. Rotating Stall Inception	71
3.3. Conclusions	91

A more complex configuration with a 3D non-axisymmetric domain is used to validate the methodology, using a generic fan-stage under forced response applying the implementation of the proposed method in the baseline flow solver Mu^2s^2T , described previously in detail. This chapter aims at identifying the viability of the proposed method to predict the unsteady flow associated with long-length scale patterns in fan-stage configurations. The stability under clean and distorted inflow is studied, retaining just a few spatial harmonics in the Fourier series. In addition, computational examples for realistic turbomachinery configurations/flow conditions, such as forced response fan-stage and clean inflow under stall conditions are given to illustrate the validity and effectiveness of the efficient approach.

The proposed method, as it has already been mentioned, is intended to be used to compute flows exhibiting large-scale instabilities and where the fundamental frequency of the problem is not known beforehand, such as rotating stall. The main uncertainty in the application of this method is to elucidate whether the reduced number of harmonics in the simulation can preclude the development of relevant rotating instabilities and the simulation resembles or not that of the full-annulus simulation. It has been shown for the first time that large disturbances, such as those encountered in a generic fan, can be accurately predicted using the proposed method with

a remarkable reduction in computational resources concerning a full-annulus simulation.

This chapter firstly describes the formation of long-wavelength disturbances as a consequence of the effect of a forced response problem due to distorted inflow operating in stable conditions. This case is analyzed in order to first validate the proposed method under a synchronous excitation and response. Finally, the stability of the fan under clean and distorted inlet flow is investigated with the proposed method. The basic assumption is that the scale of the relevant disturbances is much larger than the blade pitch, and then a few low-order spatial harmonics suffice to predict flow stall. The verification of the method is performed by comparing the proposed approach with the corresponding full-annulus analysis. The role of different spatial harmonics of the perturbations during the transient is also outlined.

3.1. Forcing Due to Inlet Distortion

Aeroengine fans often operate under distorted inlet flow conditions. This distortion is caused either by different flight operating conditions (crosswind or boundary layer ingestion) or non-uniformities due to the nacelle installation. The asymmetry of the inlet flow gives rise to non-uniform circumferential total pressure distortions which can create a strong aeromechanical response in the rotor blades. Inlet distortion can also severely compromise fan stability [12, 80, 81]. The ability to describe and quantify the flow perturbations as they progress through the fan and the compressor when subject to inlet distortion is very important. Computationally efficient tools for the analysis and design of fans and compressors able to capture this type of phenomena are highly desirable.

A large pitch ratio inlet disturbance is routinely encountered by an aircraft engine. It is known to noticeably affect the performance of the fan and thus the engine performance during flight operation. We consider a fan with high bypass ratio and subject it to a distortion in incoming total pressure. The more challenging inlet distortion is one where the deviation is once per revolution. In this situation the standard rotor-stator techniques generally do not apply and the designer must consider full 360° analysis, which is computationally prohibitive, especially early

Figure 3.1.: NASA rotor 67 semi-unstructured grid (a) and computational domain (b).

in the design process. Designers would like to have a method that both accurately resolves the effect of large scale inlet distortion, but somehow do this in an efficient and practical manner.

Numerical simulation can contribute to the design process if it can accurately predict the aerodynamic performance penalties and the loads experienced by the fan blades, thereby identifying potential problems early in the design phase. To accurately capture the required forcing function on a blade, all sources of excitation must be included in the model. This requires accurate accounting of the pressure loads on the fan from the upstream inlet distortion. The loads are inherently transient in nature, requiring solutions on the full-annulus, which is expensive and not practical early in the design cycle.

The aerodynamic performance penalties due to fan inlet distortion have been documented and published in many experimental investigations [82, 83, 84]. Fidalgo et al. [5] studied the NASA Rotor 67 transonic fan-stage operating with an inlet total pressure distortion. The numerical predictions agreed well with measured data. However, expensive full wheel, unsteady simulations were used to simulate the flow in the entire 360 degrees of the fan-stage.

Computationally efficient tools for the analysis and design of fans and compressors able to capture this kind of phenomena are highly desirable. Numerical models considering only a fraction of the annulus are not able to accurately predict the unsteady long length scale patterns of the disturbances associated with stall cells simultaneously with the short wavelength disturbances of the blade passage. The ability to describe and quantify the flow perturbations as they progress

Figure 3.2.: NASA rotor 67 meridional semi-unstructured mesh and full-annulus model.

through the fan and the compressor when subject to inlet distortion is very important as well. Some researchers [85, 86] took advantages about the periodicity of a once-per-revolution inlet distortion to use reduced geometry models in order to avoid full-annulus geometries, using a Fourier transformation 'double-passage' approach to reconstruct periodic boundary conditions [87]. Efficient reduced computational models using either phase-shifted boundary conditions [88] or linearized methods in the frequency domain [89], have been used to reduce as well the cost of the full-annulus simulations required in the presence of distorted inflow. These methods assume that the flow is periodic in time. However, operating at or near stall conditions can lead to non-axisymmetric unsteadiness with unknown frequencies. The proposed Passage-Spectral Method is aimed at capturing this unsteadiness using only a few single-passage samples.

3.1.0.1. NASA Rotor 67 Fan-stage Test Case

The methodology and flow solver have been used to study the forcing due to inlet distortion on the NASA rotor 67 fan-stage. The full-annulus of the fan geometry consists of 22 blade passages. Design parameters of the fan are listed in Tab. 3.1 at %100 of the design speed. Firstly a hybrid grid in the blade-to-blade plane with 118 points around the airfoil and 24 quad layers in the boundary layer has been constructed, next 151 layers in the span-wise direction are extruded and 12 points in the tangential direction. To accurately resolve tip leakage flow 6 mesh layers

Number of rotor blades	22
Rotational speed [rpm]	16043
Design mass flow rate [kg/s]	33.25
Design Pressure ratio	1.63
Nominal tip-clearance [mm]	0.61

Table 3.1.: NASA rotor 67 design operating conditions.

were included in the tip gap region in the span-wise direction. The semi-unstructured mesh [76] used to discretised the domain is shown in Fig. 3.1. The final grid consists of about 2.27×10^6 nodes per single-passage domain. The size of the full-annulus mesh is approximately 50×10^6 nodes. Figure 3.2 shows the mesh grid details of the full-annulus domain at the meridional plane. The Reynolds number of the simulations is very high ($Re \sim 10^6$) and the standard $k - \omega$ turbulence model including wall functions was used to alleviate mesh requirements in the boundary layers.

One-dimensional non-reflecting boundary conditions were imposed at the inlet and the outlet. At the exit, the radial equilibrium equation was applied to fix the pressure radial distribution. Non-slip and adiabatic wall boundary conditions were applied to the blade surfaces and the inner and outer casing walls.

Two types of simulations are used in this section. The first are full-annulus simulations which are used as a high-order model for verification purposes. The second are PSM simulations with a single spatial harmonic or ND matching that of the forcing due to inlet distortion. Figure 3.3 displays the three reduced-passages required to carry out the spatial Fourier transformation for a single spatial harmonic using the proposed method and the whole length of the computational domain.

3.1.0.2. Results

The selected operating condition in the fan map is represented in Fig. 3.4 as point A ($\square$), near the peak efficiency. Steady simulations have been conducted to assess the position in the fan map of the initial condition for the unsteady simulations and to perform a basic validation of the code. The computed speedlines are shown in Fig. 3.4. The steady solution at 100%

Figure 3.3.: NASA rotor 67 full computational domain with 3 reduced-passages ($M = 1$) approximation.

Figure 3.4.: NASA rotor 67 speedline from PSM simulations.

Figure 3.5.: Total pressure distribution at the fan inlet in the stationary frame of reference.

of the design speed with uniform inlet stagnation pressure at a near peak efficiency operating point is used as the initial solution for the unsteady simulations. A fan blade subjected to inlet distortion usually sees a circumferential variation in inflow conditions. There are different types of inlet distortions flow problems. In general, the distortion is a variation in the inflow conditions, for example a total temperature, total pressure or flow angle asymmetry. The total pressure profile at the inlet is modified by superimposing a steady sinusoidal perturbation in the azimuthal direction (see Fig. 3.5) creating a forcing on the rotor due to this inlet distortion. The P_0 at the inlet can be expressed as $P_0 = \bar{P}_0 + \Delta P_0 \sin \theta$. In this case, the amplitude of the inlet stagnation pressure at the inlet is $\Delta P_0 = 5kPa$, which is similar to that reported in [12]. The effect of varying the inlet stagnation pressure on the fan is investigated next with the present method.

Unsteady simulations have been conducted using 400 time steps per rotor revolution (22 blade passing periods). The temporal resolution was chosen based on a study performed for another more complex case and which will be described later on. The number of pseudo time steps per physical time step was set to 60, which is large enough to achieve the convergence criterion of the inner iteration of the dual time step. The full-annulus simulations are compared against PSM solution using just 3 reduced-passages to approximate the solution of the 1st spatial harmonic, as shown in Fig. 3.3.

The goal is to obtain a time-accurate forced response signal in the rotor, capturing the low-frequency effect of the steady inlet distortion. This once per revolution variation gives rise

Figure 3.6.: PSM solution with $M = 1$ of the total pressure contours at 0.5 axial chords downstream of the rotor trailing-edge (see Fig. 3.9) at two different time instants for the forced response distortion case.

Figure 3.7.: Axial location of screenshots shown in Fig. 3.6.

Figure 3.8.: Comparison of blade loading at 80% span between the PSM with a single harmonic (dashed lines) and full-annulus solutions ($\circ$) at three different time instants over a cycle of the inlet disturbance. The clean inlet distribution ($\diamond$) is added as reference.

to a varying incidence of the rotor blade along with its rotation. It is important to highlight that this problem could have been addressed using phase-lagged boundary conditions since the frequency of the perturbation is known. A related implication of the distortion long circumferential length scale is that unsteadiness in the rotor blade relative frame can be neglected [6]. The importance of unsteady effects is represented by the reduced frequency, St , which is the ratio between the residence time and the characteristic time. For distortions whose characteristic length scale equal to the rotor circumference, as shown in Fig. 3.5, can be approximated in terms of the rotor geometry as

$$St = \frac{\Omega_{rotor} c_x}{V_x} \quad (3.1)$$

where c_x is the mid-span axial chord of the rotor, V_x is the characteristic axial velocity at the outlet and Ω_{rotor} is the angular velocity of the rotor. For the cases presented here $St \simeq 0.06$ and the flow can be considered quasi-steady.

Figure 3.6 displays the total pressure contours one chord downstream of the rotor for the initial and the final periodic state. The solution has been obtained with the proposed method using a single spatial harmonic. The full-annulus solution is reconstructed interpolating the three reduced-passages using the computed Fourier coefficients. The aim is to compare the reduced

Figure 3.9.: NASA rotor 67 semi-unstructured grid and numerical probe location.

Figure 3.10.: Spatial Fourier decomposition of the absolute total pressure near the outer casing 0.5 axial chords downstream of fan trailing-edge (see Fig. 3.9) in the rotor frame of reference non-dimensionalised with the inlet total pressure amplitude, ΔP_0 .

geometry model (PSM) with full wheel transient predictions, thereby demonstrating the efficiency of the proposed method both in terms of accuracy and solution speed-up.

Figure 3.8 shows the variation in blade loading at 80% span of the PSM solution over a cycle of the inlet disturbance for three different time instants of the disturbance cycle ($t/T \approx 3.0, 3.2, 3.5$), where T is the time required to complete a full revolution. Strong unsteady fluctuations on the blade surface are observed. This can be fully appreciated by noting the differences between the static pressure distribution along the blade with respect the pressure distribution obtained for a clean inlet (see diamond symbols ($\diamond$) in Fig. 3.8). The location of the shock wave position noted by a sudden increase in the static pressure is seen to move upstream on the blade surface as a high-pressure wave reaches the leading-edge ($t/T \approx 3.0$), while the presence of a low-pressure region at the leading-edge moves the shock wave downstream ($t/T \approx 3.2, 3.5$). The PSM solution compares very well at the three different time instants with the full-annulus solution.

	<i>PSM</i> ($M = 1$)	<i>FULL</i>
Number of points	6.8×10^6	50×10^6
Number of GPUs (GTX 1080 Ti)	2	16
Wall clock time per revolution (h)	1.5	1

Table 3.2.: Computational cost of different numerical approximations for NASA rotor 67 case under distorted inflow conditions for a mesh of 2.27×10^6 nodes per passage.

Figure 3.10 compares the total pressure signal in a numerical probe near the outer casing, see Fig. 3.9, in the relative frame of reference, 0.5 axial chords downstream of fan trailing-edge obtained by three different simulations: the full-annulus approach, and the PSM with 1 and 3 harmonics. Starting from the solution corresponding to the clean inlet, and imposing the distortion, we can see that after approximately 4 rotor revolutions the solution becomes periodic in time exhibiting a synchronous excitation. The PSM with $M = 1$ accurately represents the forcing due to the inlet distortion downstream of the fan, as shown in Fig. 3.10 (a), for the time evolution of the first harmonic of the absolute total pressure signal in the frame of reference of the rotor. As could be foreseen, the largest disturbance is associated with the first harmonic, which is the one imposed at the inlet. It can be concluded that in this case a single spatial harmonic suffices to reproduce the full-annulus solution, with the corresponding reduction in computational resources.

In this particular case the ideal speed-up factor of the PSM, using Eqn. 2.13 with $M = 1$ and $N_B = 22$, is $R = 22/3 \simeq 7$. The actual computational effort of the two methods is shown in Tab. 3.2. It can be show that the overhead of the method is negligible.

3.2. Rotating Stall Inception

Reduced flow rates give rise to flow separation and stall conditions, resulting in large-scale fluctuations originated by one or more stall cells propagating at an unknown fraction of the rotor speed. Numerical models considering only a fraction of the full circumference are not able to accurately predict the unsteady long-length scale patterns of the disturbances associated with stall cells simultaneously with the short wavelength disturbances of the blade passage. The

Figure 3.11.: Schematic rotating stall process.

large unsteady structures of stalled fan-stages present a computational and numerical challenge, as full-annulus simulations and long domains are often required [11, 12].

The methodology and the flow solver have been used to predict the initiation of rotating stall, with clean and distorted inflow, on the NASA rotor 67 fan-stage. The formation of long-wavelength disturbances and short wavelength inception on a fan-stage under rotating stall conditions with a single and few spatial harmonics are analysed with the proposed method. A short wavelength disturbance, once formed, can create long-length scale rotating stall patterns within a few rotor revolutions with a circumferential velocity of a fraction of the rotor speed. This, of course, involves large-scale unsteady separations. These, if the process continues, are followed by the precessing of low velocity, long length scale (in the azimuthal direction) low (negative) velocity zone(s) around the annulus, as sketched in Fig. 3.11. The stall unsteadiness is then characterised by high-frequency fluctuations rotating at a fraction of the rotor speed Ω caused by perturbations with a length-scale of the order of a blade pitch, and long-wavelength rotating structures with scales of the order of the whole circumference.

A duct of approximately $0.3D$ and $0.25D$ has been added upstream and downstream respectively of the rotor to ease fan stability analysis although it is acknowledged that the stability margin depends on this location [12]. It was concluded that as the exit duct becomes longer, the stall margin loss of the blade increases. Figure 3.12 displays the whole length of the computational domain and the three reduced-passages required to carry out the spatial Fourier transformation for a single spatial harmonic. The final grid consists of about 3.53×10^6 nodes per single-

Figure 3.12.: Long NASA rotor 67 full computational domain.

passage domain. The size of the full-annulus mesh is approximately 78×10^6 nodes.

Design parameters of the fan are the same than in the previous section, see Tab. 3.1 at %100 of the design speed. The solutions obtained by the present method for clean and distorted inlet flow conditions is compared against full-annulus transient predictions. The goal is to use a few number of reduced-passages to capture the large-scale non-axisymmetric perturbations of the flow during the stall process, filtering the highest order harmonics which are not of interest. The basic assumption is that the scale of the relevant disturbances is much larger than the blade pitch, and then a few low-order spatial harmonics suffice to predict flow stall. The method is not only an efficient computational tool avoiding expensive full-annulus simulations but provides valuable information as well about the most destabilizing harmonics during the stall process.

Figure 3.13.: Entropy distribution of tip region (99% span) after 16 rotor revolutions.

3.2.1. Clean Inflow

The first verification case consists of the NASA rotor 67 near stall conditions with uniform inlet conditions. This point is represented in Fig. 3.4 as point B ($\circ$), near operating stall conditions. The steady solution with uniform inlet stagnation pressure near stall operating conditions is used as the initial solution for the unsteady simulations. When the mass flow is reduced at constant rotational speed, the nominally uniform still unstalled flow becomes unstable and develops a stall inception which is significantly non-uniform in the circumferential direction. The flow becomes inherently unsteady and steady simulations can not predict this unstable behaviour.

Unsteady simulations are conducted then with 400 time steps per rotor revolution (22 blade passing periods). The number of pseudo time steps per physical time step was set to 60, which is large enough to achieve the convergence criterion of the inner iteration of the dual time step. The full-annulus simulations are compared with a single harmonic PSM solution, as shown in Fig. 3.3. A static pressure exit boundary condition was chosen that resulted in a near stall operating condition. The stall onset is sought gradually increasing the exit static pressure at constant rotor speed.

3.2.1.1. Results

Figure 3.13 compares the entropy snapshots in the tip region (99% span) at the same time instant of the PSM (a) and the full-annulus (b) computations after 16 rotor revolutions. At this time instant the fan is operating at point B (○) in Fig. 3.4, which basically means that the pressure ratio of the fan is nearly the initial one but the mass flow has been substantially reduced. Entropy contours are a good indicator of separated vortical structures. A spillage of tip leakage flow near the tip region (99% span from hub) occurs around the leading edge, as previously reported by other authors [16]. The first spillage changes the load distribution at the trailing edge region causing the propagation of the spillage along the blade row, increasing the amplitude of the first circumferential mode, resulting in a short-length scale multiple-cell pattern of the order of one rotor circumference which can be seen in Fig. 3.13. The first spatial harmonic of the perturbation gradually increases with time across the whole domain for both solutions. The stall cell spans multiple rotor passages as it rotates around the annulus. The flow-field solution obtained by the PSM, see Fig. 3.13 (a), is in good agreement with that computed using the full-annulus domain, see Fig. 3.13(b).

Figure 3.14 displays a snapshot of the relative axial Mach number at the mid-chord of the rotor blade at two different time instants (16 and 20 rotor revolutions), for the full-annulus solution and the one obtained with a single spatial harmonic (3 reduced passages). It can be seen that the PSM is able to reproduce the complex unsteady behaviour of the full-annulus simulations at both time instants. The creation of a stall region at the rotor tip can be clearly distinguished. After 16 rotor revolutions, as shown in Figs. 3.14 (a) and (b), a sinusoidal type disturbance with a wavelength of the order of one rotor circumference is developed, as it has been previously outlined for the entropy distribution at the same instant time, as shown in Fig. 3.13. This perturbation with respect the base flow is localized in the rotor blade tip region and evolves into stall cells. It is shown that a long-wavelength pattern, of the size of the whole circumference, and short wavelength disturbance, of the order of the pitch, are triggered. The short wavelength is absorbed within the passage mesh while the long-wavelength is represented using the Fourier series. This can be observed after 20 rotor revolutions, as shown in Figs. 3.14 (c) and

Figure 3.14.: Snapshots of the relative Mach number at the mid-chord of the fan rotor during stall inception at "Near Stall" operating conditions, see point B (○) in Fig. 3.4.

Figure 3.15.: Comparison of the relative isentropic Mach number at 80% span between PSM ($M = 1$) and full-annulus solutions subjected to rotating stall after 16 rotor revolutions. The "Near Stall" steady distribution ($\diamond$) is added as reference.

(d), where both solutions exhibit a completely detached flow. The low-speed region, initially confined to the tip region, has grown radially inwards and circumferentially becoming a single stall cell. The contours of relative Mach number at the breakdown point condition show that stall propagates in counter rotation, from one side to its opposite, where the blade passages remain unstalled, leading us to conclude that the first harmonic perturbation has the most destabilizing effect.

In order to perform a more quantitative comparison between the PSM and the full-annulus solution, the instantaneous relative isentropic Mach number distributions at 80% span after 16 rotor revolutions on the three different blade samples are shown in Fig. 3.15. The Mach number distributions in the three blades are significantly different, which is a clear indication that the level of unsteadiness at this time instant is very high. However, the PSM with a single harmonic (continuous lines) predicts very well the distributions obtained by the full-annulus simulations (symbols) in all the blades, which are uniformly spaced in the azimuthal direction by 120° . This is a clear indication that in this case, and at least up to this time, a single harmonic suffices to accurately represent the solution.

Figure 3.16 displays the relative isentropic Mach number contours on the blade surface for

Figure 3.16.: Relative isentropic Mach number contours of the three reduced-passages used in PSM ($M = 1$) solution subjected to rotating stall inception after 16 rotor revolutions.

the three different reduced-passages samples after 16 rotor revolutions obtained by the PSM. The same variations seen in Fig. 3.15 are present in the whole surface. The first blade sample ($\theta = 0^\circ$) is stalled while the second ($\theta = 120^\circ$) is beginning to stall whilst the third blade sample ($\theta = 240^\circ$) is attached.

Stall cell speed. The propagation speed of the stall cells can be determined by examining their azimuthal location at different time instants. Figure 3.17 displays the instantaneous axial velocity of a point of the first reduced-passage located upstream of the rotor in the tip region during three rotor revolutions. It is concluded that the rotational speed associated with this disturbance is about 40% of rotor speed counter to the rotation, since the rotating stall cells, highlighted with a circle in Fig. 3.17, rotate approximately 145° in one rotor revolution in the absolute frame of reference. The rotational speed of the stalled pattern in the PSM solutions is in good agreement with that obtained with the full-annulus domain, and the fluctuation amplitudes are also comparable. These results can be compared with the ones obtained experimentally by Inoue et al. [90] and numerically by Hah et al. [91]. Their conclusions indicate that the stall process takes between one and two rotor revolutions to develop from the stall inception point, Figs. 3.14 (a) and (b), to the breakdown flow conditions, Figs. 3.14 (c) and (d), and the stall cells rotate at about half of the rotor speed.

Figure 3.17.: Propagation of rotor upstream axial velocity disturbance at 0.5 axial chords of the fan trailing-edge and 99% span from hub for the PSM solution with $M = 1$.

Good agreement is found between PSM with a single spatial harmonic and full-annulus solutions for a stalled fan-stage under the effect of clean inflow. The stall inception and the transient behaviour of the rotating stall cell is accurately predict by the proposed method in comparison with full-annulus results, with the corresponding savings in computational resources. The speed-up factor R , as outlined in Eqn. 2.13, of the PSM ($M = 1$) solution with respect the full-annulus is $R = 22/3 \simeq 7$.

It is concluded that the method is able to capture the non-cyclic symmetry of the instability and the propagation speed of the stalled cells even with a single harmonic, indicating that the solutions are weakly dependent with the retained number of harmonics. However, this issue will be addressed in more detail in the next section.

3.2.2. Distorted Inflow

A physically induced irregularity in the approaching flow can lead to a premature dynamic separation of the blades propagating along the compressor circumferential direction in the sense of rotor rotation. This instability destroys the cyclic symmetry of the flow since typically a

Figure 3.18.: Top: Comparison of the full-annulus and the PSM transient spatial Fourier decomposition of the total pressure in a probe near the outer casing at 0.5 axial chords downstream of fan trailing-edge (see Fig. 3.9) non-dimensionalised with the inlet total pressure amplitude, ΔP_0 . Bottom: non-dimensional relative angular phase velocity of the spatial harmonics, Ω_m/Ω_{rotor} , at the final saturated state.

fraction of the annulus contains stalled airfoils while the remaining exhibits attached flow. This operating condition represents a purely unsteady behaviour with rotating stall structures as it has been reported experimentally [18] and numerically using full-annulus simulations [81]. The sinusoidal shape of the inlet distortion profile promotes the development of transient low-order spatial harmonics. This is different from other types of inlet distortion experiments [10, 12] where the blockage is represented by a sector with a constant low pressure. In these cases, due to the presence of a discontinuity at the inlet, the signal contains high-order spatial harmonics as well.

A comparison between full-annulus simulations and the PSM is carried out next to assess the accuracy of the method under stall conditions for two different inlet distortion amplitudes and the same nodal diameter, $ND = 1$. The static exit pressure was chosen in this case such as the operating point is near stall conditions, corresponding to point B (○) in Fig. 3.4. The steady solution in the absence of inlet distortion is used as the initial solution of unsteady simulations. Pressure signal are measured in a numerical probe located at near the outer casing at 0.5 axial chords downstream of fan trailing-edge, as previously defined in Fig. 3.9.

Figure 3.19.: Axial velocity contours at 0.5 axial chords downstream of the fan trailing-edge (see Fig. 3.9) after 20 rotor revolutions for the low total pressure amplitude distortion case at the final saturated state.

3.2.2.1. Low total pressure amplitude distortion.

The amplitude of a sinusoidal total pressure disturbance at the inlet is set to $\Delta P_0 = 5kPa$. The inlet total pressure profile is the same as the previous section for the NASA rotor 67 fan-stage under forced response distortion, which corresponds to a sinusoidal total pressure distortion at the inlet with $ND = 1$, as shown in Fig. 3.5, but modifying the operating point to near stall conditions, see point B (○) in Fig. 3.4.

Harmonic content of the solution. The PSM with $M = 3$ and the full-annulus solutions have been Fourier decomposed in space to investigate the harmonic content of both solutions. Figures 3.18 (a), (b) and (c) show the time evolution of the first three harmonics of the axial velocity in a numerical probe near the outer casing, see Fig. 3.9. It was found that the flow takes about 8 rotor revolutions to reach a converged quasi-periodic state. The time is long enough to let the inlet disturbance travel along the fan-stage and initiate a quasi-stable rotating stall process. It can be seen that the 2^{nd} harmonic is the most unstable one. According to the results, it can be argued that the excitation of this second spatial harmonic takes place through a non-linear mechanism since only the first spatial harmonic is perturbed at the inlet, since the perturbation corresponds to a $ND = 1$ azimuthal distribution. The amplitude of the 2^{nd} harmonic is greater

than that of the first while the amplitude of the third spatial harmonic is much smaller than that of the first and the second. These results are consistent with Fig. 3.19, where two regions of low velocity can be seen, which is a clear indication that the second harmonic is the most important one.

To further study this statistically converged quasi-periodic state the time-varying amplitude of each spatial harmonic has been Fourier Transform in time. To obtain a clean signal the initial part of the transient has been discarded in the analysis. The result is presented in Figs. 3.18 (d), (e) and (f) where the horizontal axis represents the angular velocity of propagation of the harmonic relative to the rotor, $\Omega_m = \omega_m/m$, non-dimensionalised with the rotor speed, Ω_{rotor} .

Firstly, it is seen that the agreement between the two methods is good and that the PSM simulation can capture the instantaneous full-annulus solution at the monitoring point very well. The method can capture as well the time spectrum created during the non-linear process associated with the stalled flow, as shown in Figs. 3.18 (d), (e) and (f). It is important to recall that the spectrum is rich in frequencies as a consequence of the interaction of the stalled cells with inlet excitation.

Interestingly, Fig. 3.18 (e) shows that the propagation velocity of the 2nd harmonic of the stall cell relative to the rotor is about 14% shaft speed which approximately matches the results presented by Kuen-Bae et al. [92] for a low-speed fan. This harmonic has the largest contribution to the stall cell. This propagation velocity can be tracked as well in 1st harmonic and in a much lesser extent in the third harmonic. A detailed discussion of this solution is outside of the scope of this research.

The purpose of this case was to prove that the proposed method provides a similar solution as the full-annulus case, both in terms of spatial and temporal harmonics, in a flow with a richer harmonic content than the previous stable case which leads to a synchronous forced response problem. In this operating point and with the selected level of forcing the solution is unstable but with a limited amount of unsteadiness and a relatively simple spectrum. It is important to recall that in this case, only 7 out of the 22 passages of the rotor were included in the simulation.

Full Annulus vs PSM. Figure 3.19 shows the contours of the axial velocity of the PSM and

the full-annulus solution after 20 rotor revolutions. This snapshot is representative of the character of the solution at other time instants due to the quasi-periodic state of the solution once the flow reaches this stall condition. Comparing both solutions, and taking into account the good agreement of the time evolution of the first three harmonics, we can conclude that the proposed Passage-Spectral Method can accurately predict the time evolution of the full-annulus geometry with only three spatial harmonics ($M = 3$). A two-lobe pattern of low total pressure can be clearly distinguished in Fig. 3.19, which confirms that the 2nd harmonic has the largest contribution to the flow solution. The 1st harmonic is directly perturbed by the distorted inflow and other frequencies are also created. However, the contribution of the 3rd spatial harmonic to the unsteady behaviour is almost zero, as well as that of the higher harmonics ($m > 3$) (not shown in this study since they are negligible).

3.2.2.2. High total pressure amplitude distortion.

The amplitude of the total pressure disturbance at the inlet is increased now from $\Delta P_0 = 5kPa$ to $\Delta P_0 = 10kPa$ at the same near stall operating conditions (point B (○) in Fig. 3.4). In this case, the compressor is expected to be more unstable and the flow exhibits a denser frequency spectrum. Figure 3.20 shows the time evolution of the mass flow for the PSM and the full-annulus solutions during the stall process, departing from the same initial solution. It can be concluded that there is a good agreement between both solutions and that the decrease of the mass flow after two rotor revolutions is well captured. The low total pressure region at the inlet promotes the development of strong stall inception, where low-velocity regions near the tip are created.

Time-step Sensitivity. The sensitivity of the solution to the time resolution for this complex case is investigated using different numbers of time steps per rotor revolution. Simulations with 200, 400 and 4,000 time steps per rotor revolution have been carried out, both for the PSM with three harmonics ($M = 3$) and the full-annulus domain. The convergence study is displayed in Fig. 3.21 where the mass flow at the exit is plotted as a function of the number of rotor revolutions. It is observed that for 200 time steps per revolution, the time history of the PSM,

Figure 3.20.: Mass flow history at the outlet as a function of the time resolution.

Figure 3.21.: Time required to achieve a stall mass flow of reference ($\dot{m}_{ref} \simeq 31.1 \text{ kg/s}$) as a function of the time resolution for the PSM with $M = 3$.

and the full annulus are different. Furthermore, both time evolutions are different from that obtained with 400 and 4,000 time steps per revolution. The latter case is deemed to be over resolved and therefore since the difference between both transients is small, 400 time steps per revolution was adopted as the baseline time resolution in this work. It is important to highlight also that if the time resolution is high enough, the time histories of the PSM and the full annulus domain are nearly identical.

Figure 3.21 displays the physical time, measured in terms of rotor revolutions, at which a reference mass flow, $\dot{m}_{ref} \simeq 31.1 \text{ kg/s}$, is obtained. It can be seen that for more than 400 time steps

per rotor revolution the sensitivity is low. However, the stall margin is greatly influenced by the size of the time step [80]. The use of large time steps over-predicts the stall margin delaying the stall inception at a given mass flow of reference. This can be understood by noting that increasing the time step filters out the physical solution preventing the appearance of high-order modes. On the view of this study, it is concluded that, at least in this case, 400 time steps per time revolution seem to be a proper trade between accuracy and computational cost.

Harmonic content of the solution. The full-annulus solution has been Fourier decomposed in space to investigate the harmonic content of the solution. Figure 3.22 shows the time evolution of the first ten harmonics of the axial velocity. The tracking point is located in the rotor frame of reference near the casing at 0.5 axial chords downstream of the rotor trailing-edge. It can be seen in Figs. 3.22 (f), (g), (h), (i) and (j), that the high-order harmonics ($m > 5$) have a small contribution to the unsteady behaviour of the full-annulus solution. This is an indication that a similar level of accuracy could be achieved with a reduced number of spatial harmonics.

Similar conclusions were drawn by He [62] who quantitatively captured pairs of contra-rotating vortices in non-axisymmetric flow in a rotor/stator compressor cavity with a low number of spatial harmonics using a similar approach. However, this case is more involved since the fan separated flow gives rise to a very rich harmonic content.

A second observation is that all the high-order harmonics are triggered simultaneously, approximately at rotor revolution 5. At this stage the flow has already breakdown and large stalled cells are present in the flow-field. Figure 3.23 displays a snapshot of the solution after 2 and 6 rotor revolutions. We can see that for this high amplitude inlet total pressure distortion profile, $\Delta P_0 = 10kPa$, the solution has a greater number of perturbed harmonics (see Fig. 3.22) than the case with a low amplitude distortion, $\Delta P_0 = 5kPa$ (see Fig. 3.18).

Full-annulus vs PSM. Figures 3.22 (a), (b) and (c) compare the temporal evolution of the first three spatial Fourier harmonics of the full-annulus solution with the PSM. During the first two rotor revolutions, the first harmonic of both the PSM and full-annulus solutions, as shown in Fig. 3.22 (a), experiences a steep increase in its amplitude, which indicates a rapid development of the non-axisymmetric flow in this region. This is associated with the effect of the inlet

Figure 3.22.: Full-annulus Fourier decomposition of the relative axial velocity near the outer casing at 0.5 axial chords downstream of the fan trailing-edge (see Fig. 3.9) non-dimensionalised with the inlet reference axial velocity for the high total pressure amplitude distortion case. Solid lines: full-annulus solution. Dashed lines: PSM with $M = 3$.

Figure 3.23.: Axial velocity contours at 0.5 axial chords downstream of the fan trailing-edge (see Fig. 3.9) at two different time instants for the high total pressure amplitude distortion case at the final saturated state.

perturbation which contains solely the first harmonic. This vortical perturbation created by the inlet stagnation pressure propagates along the duct and impinges the fan blade, creating a perturbation which contains mainly the first nodal diameter. Figure 3.22 (a) shows that the 1st harmonic evolution of the PSM and the full-annulus solutions are nearly identical up to the fifth rotor revolution.

After approximately 4 rotor revolutions, the third harmonic, as shown in Fig. 3.22 (c), is triggered and experiences a rapid exponential-like growth. It is speculated that this is probably the most unstable mode. This destabilizing effect results in a three-lobe rotating pattern in the absolute frame of reference, as can be seen in Figs. 3.23 (c) and (d) after 6 rotor revolutions. The three-lobe pattern is more clearly seen in the PSM than in the full-annulus solution, which is richer in harmonic content.

Once the stall process is triggered, the first harmonic ($m = 1$) is shown to be persistent in time and coexists with the third harmonic ($m = 3$). This can be interpreted by noting that the 1st harmonic is continuously excited by the inlet and therefore must be always present in the simulation. However, the 3rd harmonic is the most dominant mode after 5 rotor revolutions at the final saturated state. This is more true in the PSM solution.

The evolution of the 2nd and 3rd harmonics of the PSM and the full-annulus solutions, although qualitatively similar, are not exactly the same. The initial creation of a third harmonic perturbation, and its subsequent growth, is a non-linear phenomenon since in the initial solution the third harmonic is essentially zero. Figure 3.22 (b) shows that the second harmonic is non-linearly created after the first one, approximately after 1.5 revolutions. Then the third harmonic, the most unstable one, is triggered and begins to grow quickly. This sequence of events can be seen both in the full-annulus and the PSM simulations, what is a clear indication that the PSM is qualitatively correct.

Figures 3.22 (d) and (e) show that the fourth and fifth harmonics are not negligible after the fifth revolution. This is the ultimate reason why, for long computational times, the solution of the PSM differs from that of the full-annulus. The non-linear process which takes place after the flow breakdown cannot be reproduced just with three harmonics since the real solution is

filtered out. However, the instability of the fan, which develops during the early stages of the process, is well captured as well as a good agreement of the mass flow time evolution (see Fig. 3.20). The method with $M = 3$ can reproduce the unsteadiness associated with the mass flow evolution during the stall process despite of filtering higher harmonics.

Flow description. Figure 3.20 shows that both mass flow time histories drop sharply after the second rotor revolution, which is caused by the effect of the low-pressure region of the inlet distortion pattern, which tends to reduce the mass flow rate and stall the flow locally. At this instant, the stall inception near the tip region has taken place. Figures 3.23 (a) and (b) compare the instantaneous contours of the axial velocity at 0.5 axial chords downstream of the fan trailing-edge after two rotor revolutions and no significant differences can be noticed between the PSM and the full-annulus solutions. A low-velocity separation region can be identified near the tip region. The results also show an increase in the axial velocity in the hub region induced by the blockage of the tip region [12]. Both simulations exhibit a similar circumferential pattern of one rotor circumference and the corresponding non-uniformity.

Once the mass flow decreases to stall levels the non-uniformities grow, and the flow massively separates. Figures 3.23 (c) and (d) compare the axial velocity contours after 6 rotor revolutions. At this instant, the circumferential periodicity of the flow created by the rotor blade cannot be seen any longer. long-wavelength patterns in the tip region which propagate radially inwards can be distinguished. It can be noticed that the reduction in the mass flow and the increase in the magnitude of the flow disturbances happen simultaneously.

Figure 3.24 shows a snapshot of the reconstructed instantaneous static pressure contours obtained by the proposed Fourier method. Stall inception structures downward in the span-wise direction can be identified inside the blade passage and the effect of stall inception downstream of the fan. The local pressure gradient is increased on the pressure surface at the rear of the blade passage in the region of the stalled blades in Fig. 3.24, resulting in a non-axisymmetric pattern in the circumferential direction.

Figure 3.24.: Instantaneous static pressure contours of the PSM ($M = 3$) solution at 60% span from hub.

	$PSM (M = 3)$	$FULL$
Number of points	24.8×10^6	78×10^6
Number of GPUs	4	16
Wall clock time per revolution (h)	1.8	1.4

Table 3.3.: Computational time for the NASA rotor 67 unstable cases under rotating stall for a mesh of 3.53×10^6 nodes per passage.

3.2.2.3. Computational Efficiency

The computational effort reduction factor, R , of the PSM under low and high amplitude inlet total pressure distortions cases with $ND = 1$, which trigger a rotating stall inception, using three harmonics ($M = 3$) with respect the direct full-annulus solution is $R = 22/7 \simeq 3$, since we simulate 7 reduced-passages sample out of the 22 passages of the fan-stage, $N_B = 22$. Table 3.3 compares the actual computational effort required by the two methods for the stalled fan-stage simulation. The interesting point is that these simulations can be performed overnight with modest hardware requirements.

3.3. Conclusions

To the author's knowledge, this thesis presents for the first time the application of a reduced-passage method to non-axisymmetric geometries such as forced response due to inlet distortion and rotating stall conditions in a fan-stage without any assumed frequency. The NASA rotor 67 under the effect of clean and distorted inflow conditions has been used as a second verification case to demonstrate the effectiveness and viability of the Passage-Spectral Method for different operating points of the design speedline. The one per revolution disturbance created in the inlet due to the presence of a distorted inflow gives rise to a stable solution, and the sole effect of the distorted inlet is a periodic forcing in the rotor. The PSM is verified by comparing to the equivalent full-annulus unsteady solution. The PSM unsteady solution compares remarkably well with the reference solution and is able to reproduce the detailed dynamics of the shock movement. Moreover, the solution from the PSM method is also able to reproduce the efficiency, viscous effects and blade loading from the full-annulus case. The solution has been accurately reproduced by the method just retaining three reduced-passages samples with a reduction of a factor of seven in GPU time and memory. This number heavily depends on the ratio between the number of retained harmonics to the number of blades.

To study the potential of the proposed method in unstable configurations, the stall inception on the NASA rotor 67 with clean and distorted inlet flow has been investigated using the proposed method with a different number of harmonics in each case. The stability of a fan-stage under stall conditions due to clean and distorted inflow in response to a small passing distortion of the order of one rotor circumference has been identified with the initiation of stall inception with a long-length scale multiple-cell pattern, which emerged due to tip clearance vortex breakdown, using few low order harmonics solution in the circumferential direction. It has been shown that during the early stages of the process (i.e. the instability inception) the proposed method used with a reduced number of harmonics is able to reproduce the full-annulus solution accurately. However, for high inlet total pressure amplitude distortions the flow breaks down and the fan is massively separated, the harmonic content of the solution is very rich and a few harmonics do not suffice to quantitatively reproduce the long-term evolution of the flow. The method

fails to approximate spatial modes out of the range of the harmonic content resolution of the proposed block Fourier transform described in Chapter 2. This is the ultimate reason why random non-uniformities cannot be represented exactly on a reduced number of passages. It has been demonstrated that the present method can be used at least to assess fan stability even when the number of harmonics used in the PSM simulation is not enough to reproduce quantitatively the long-time behaviour of the flow which can contain a large number of harmonics. The method accurately predicts the unsteady features associated to the large scales present in rotating stall together with short wave-length disturbances with high non-linearities for a low-aspect-ratio fan just retaining a few spatial harmonics with adequate accuracy and a significant reduction in computational time and memory.

The computational efficiency of the method is very high and depends on the ratio between the number of retained spatial harmonics and the number of blades. Speed-up factors of 3 to 7 has been demonstrated. However, fan configurations do not fully exploit the method's potential due to its relatively low blade-count. It is expected that when applied to other types of blading, such as low-pressure or high-pressure compressors, the benefit will be much higher. The structure of the solution, which is naturally decomposed in spatial harmonics, is a by-product of the method which contributes to ease the physical understanding by facilitating the tracking of different harmonics. Concluding, the method is well-suited to be applied to problems with flow features much larger than the blade pitch without a beforehand known frequency.

4. Non-linear Stability Analysis of a Generic Fan

Contents

4.1. Modified Block Spectral Method	94
4.2. Inlet Total Pressure Perturbation Stability Analysis	96
4.3. Outlet Static Pressure Perturbation Stability Analysis	113
4.4. Proposed Methodology	124
4.5. Conclusions	125

This chapter aims at studying the dependence of the aerodynamic stability of fan blades with distorted inflow and potential perturbations associated with the presence of pylons as a function of the nodal diameter (non-dimensional circumferential wave-length) and amplitude of the perturbation. The explicit spatial Fourier approximation is exploited to characterise the relevance of each nodal diameter of inlet and outlet perturbations in the fan stall process. The method represents very efficiently the unsteady flow generated by inlet and outlet non-uniformities and studies the non-linear stability in a harmonic by harmonic basis. This approximation allows studying the contribution to the stall of each circumferential mode separately.

The methodology has been assessed for the NASA rotor 67. The maximum amplitude of total and static pressure distortion, in the inlet and outlet respectively, at which the compressor becomes unstable and triggers a stall process has been mapped. It has been proven that despite the complexity of a screen-induced total pressure distortion, or the number of downstream struts,

Figure 4.1.: Sketch of the location of three-reduced passages ($N_J = 3$) to represent an inlet perturbation with $M = 3$.

the only relevant parameter for the non-linear stability of the fan is the most unstable nodal diameter. The equivalence in terms of stability between realistic multi-harmonic patterns and a single harmonic distortion has been assessed. It is concluded that performing a non-linear simulation with the proper single harmonic perturbation is enough to assess fan stability. It is shown that for the NASA rotor 67 running at the nominal speed the most unstable nodal diameter is the first for both inlet and outlet perturbations.

This study not only shows a reduction in computational time to assess non-linear fan stability by a factor of seven but also creates an efficient methodology for understanding the non-linear instability of fans due to inlet and outlet distortion profiles. The proposed method can predict the unsteady flow associated with the large scales present in the stalled flow induced by inlet and outlet distortions just retaining a few spatial harmonics of the solution.

4.1. Modified Block Spectral Method

Unsteady simulations with different inlet distortion patterns have been carried out for the NASA rotor 67 fan blade to investigate its aerodynamic stability under the effect of different distortion screens. In order to do that, the reduced order Passage-Spectral Method and full-annulus rep-

representations of the fan have been used. The PSM has been slightly modified aiming at handling inlet distortion screens in a nodal diameter by nodal diameter basis efficiently without making any hypothesis about its temporal periodicity.

This method has been modified and customized for this work to study the contribution of a given single spatial harmonic M efficiently, by approximating the solution with only three reduced-passages samples, $N_J = 3$, spanning over a sector of size $2\pi/M$, being N_J the number of passages retained in the Fourier transform.

$$\hat{\mathbf{U}}_i^m(\mathbf{x}, t) = \frac{1}{N_J} \sum_{j=1}^{N_J} \mathbf{U}_i^{(j)}(x, r, \theta_j, t) e^{-Im\theta_j} \quad (4.1)$$

The code is based on the virtual existence of a Fourier transformation of the homologous nodes of the different passages, $\mathbf{U}_i^{(j)}$, and the description of the flow in terms its Fourier coefficients in Eqn. 4.1 where now

$$\theta_j = \frac{j-1}{MN_J} 2\pi \quad (4.2)$$

The purpose of using a Fourier modeling of the spatial flow-field is to use a truncated Fourier series of the “exact” solution and retain only the most representative harmonics. The underlying hypothesis is that the wavelength of the circumferential non-uniformities is long, and therefore, a few harmonics suffice to represent the long wave-lengths. Short wave-length variations are retained within the blade passage mesh. Obviously, the Fourier transform of all the mesh nodes is impractical and does not need to be computed. Since the numerical scheme is local in nature, only the nodes located near the periodic boundaries need to be informed and corrected with the global information carried by the Fourier coefficients. In this fashion, only a small subset of nodes need to be Fourier transformed.

If only a single arbitrary harmonic of the perturbed flow is retained then just three equally spaced passages need to be computed to reconstruct the mean value and the harmonic, as explained in Chapter 1. The main idea of the modified method is sketched in Fig. 4.1 where it can be seen that the 3rd harmonic ($M = 3$) of the circumferential variations of the flow-field is reconstructed using only three passages (in dark gray) in a 120° sector to approximate the flow due to an

excitation containing just the 3rd nodal diameter. The solution in the rest of the passages (in light gray), inside and outside of the 120° sector, is interpolated using a Fourier reconstruction using solely the information of the three single-passage domains and the periodicity condition in the studied 120° sector.

4.2. Inlet Total Pressure Perturbation Stability Analysis

Aircraft designs wherein the propulsive turbomachines are partially or fully embedded within the airframe present serious challenges about the aerodynamic performance of the compression system. These problems are associated with the response to the distorted inflow caused by the nacelle and intake geometries, reducing the aerodynamic stability of the engine. Inlet distortion refers then to the non-uniform flow created upstream of the fan, or any other row of airfoils, which can detrimentally affect its performances in terms of efficiency or pressure rise. The performance and aerodynamic stability of fan blades operating in a circumferentially non-uniform inlet total pressure distribution was identified a long time ago as a key issue in the development of gas turbine engines and a serious concern in the design of turbofan engines [93]. Besides, to understand the effects of inlet distortion on the overall compression system and the engine, fan and compressor designs can be improved by specifically analysing the influence of inlet distortion and how different patterns of total pressure distribution affect the stability of the compression system [94, 95]. Non-uniform total pressure distribution at the inlet of the fan reduces its stable operating range and potentially creates forced response and mechanical integrity problems on the blades. This interaction and the effect of different inlet total pressure distributions become more important for the lip intake design of the new generation of short intake engines. Page et al. [96] calculated the stall inception process with different inlet distortions generated by curved intake ducts. The results provided evidence that the low momentum flow in the tip region leads to damaging instability. As a consequence, the relevance of studying fan stability under the effects of distorted inflow has been universally acknowledged.

The study of fan stability under the effect of specific profiles mimicking realistic environments,

such as those caused by boundary layer ingestion [6], flow separation in the inlet duct [81] or sudden manoeuvres of the aircraft [12], is an understudied part of the engine/airframe interaction and an active area of research [97]. Experimental and numerical investigation by Hah et al. [98] have shown that the inlet total pressure distortion increases aerodynamic losses when a strong unsteady interaction between the passage shock and the blade boundary layer develops. Further, experimental research on an S-duct was completed by Ferrar et al. [99] and various mass flow rates tested. Results were similar to numerical simulations where curvature in the duct affects the static pressure gradient due to flow separation. The majority of recent works have focused on analysing the effect on the stability of the distortion through a compressor stage. Zhang et al. [80] showed that for an inlet distortion with fixed strength, the stall margin loss increases as the rotational speed decreases, but very little attention has been paid to the effect of different circumferential extents and amplitude of the distortion pattern on the stability of the system.

A large number of experiments have been performed to explore the relationship between inlet distortion and compressor performance [100, 101, 102]. However, Computational Fluid Dynamics (CFD) provides an alternative method to determine the stability loss of a fan blade due to complicated inlet distortion patterns. Many researchers have assessed the effect of inlet distortion to understand, analyze, and predict its impact on the operation of an engine [103, 104, 105]. The common characteristic of these numerical strategies based on URANS simulations is their high computational cost. Modelling the aerodynamic effects using full-annulus transient simulations though feasible [5, 92], is still too expensive for routinely used in design offices.

As an alternative to harmonic balance [106] or time-domain Fourier [61] reduced order methods, a time-marching Passage-Spectral Method is proposed in this study to reduce the computational cost of these simulations and simplify the study of fan-stage stability under the effect of distorted inflows without making any hypothesis about its temporal periodicity. An existing parallel CFD code [107, 108] has been modified and customized to study the aerodynamic response of turbomachinery rows due to distorted inlet flows by separating the contribution of each nodal diameter at the inlet. The methodology outlined in this thesis has been further modified to deal

efficiently with a single spatial harmonic and then applied to assess the effect of inlet distortion on the aerodynamic stability of the fan. In this model just three reduced-passages are retained.

This Chapter aims to prove that despite the complexity of screen-induced total pressure distortion the most relevant characteristic to parametrize fan stability is the most non-linearly unstable nodal diameter. In this section firstly, the non-linear stability of each nodal diameter at the inlet is investigated as a function of the perturbation amplitude taking advantage of the use of the novel method. Next, the methodology is applied to a generic transonic fan blade with different total pressure distortion screens. Finally, an efficient methodology to assess fan stability on the presence of inlet distortion is proposed.

4.2.1. Fan-Stage Stability

This section aims at studying the non-linear aerodynamic stability of a generic fan blade operating at a constant speed under distorted inflow. The idea of characterizing the effect of inlet distortion on fan stability using the Fourier coefficients of the inlet perturbation was pursued by Peterson et al. [109], but their studies were inconclusive. Here this same idea has been sought but more systematically and with different numerical tools.

Intake distortion is parametrized in terms of its nodal diameter and amplitude of the distortion, where two stable and unstable regions will be distinguished as a function of the two studied parameters. The incoming and the resulting flow disturbances are usually dominated by long circumferential wave-lengths (i.e.: much longer than the blade pitch) which makes the Passage-Spectral Method (PSM) well suited for its analysis since it avoids the use of full-annulus simulations. Moreover, the method allows investigating the independent contribution of the different NDs efficiently. It is important to point out that this is not the case in non-linear full-annulus simulations since when the perturbations are large, and the linear stability regime is by-passed, the non-linear interaction amongst the different NDs is unavoidable. However, the PSM method intrinsically works in a nodal diameter basis and the modal content of the solution can be tailored and filtered out. In other words, the PSM naturally allows to investigate the contribution to the non-linear stability of the system of each spatial circumferential mode,

whereas full wheel analyses not since the flow non linearity can gives rise to complex non-linear interactions.

In this regard, the method has been adapted to simulate an arbitrary single ND perturbation using solely three reduced passages to fit the desired wave-length. This implies that, by construction, the non-linear interaction amongst different NDs is excluded. The method allows to study the contribution to the stability of the different spatial harmonics separately, just retaining three reduced-passages over a sector of $2\pi/ND$ degrees (see Fig. 4.1), and simulate the effect of this single harmonic filtering the non-linear contribution of the rest. Two types of simulations are used in this section.

The NASA rotor 67, which is a low-aspect-ratio fan, has been chosen as the test vehicle as in the previous Chapter 3. Design parameters and details of the mesh grid (see Fig. 3.1 and Fig. 3.2) have already been described in Tab. 3.1 at %100 of the design speed. The full length domain was outlined in Fig. 3.12. The selected operating condition in the fan map for the steady simulation at 100% of the design speed with uniform total pressure at the inlet is represented in Fig. 3.4 as point A ($\square$) at a near peak efficiency operating point, which is used as the initial solution for the unsteady simulations. Unsteady simulations have been conducted using 400 time steps per rotor revolution. The number of pseudo time steps per physical time step was set to 60, which is large enough to achieve the convergence criterion of the inner iteration of the dual time step. The base flow is uniform and axial at the inlet, but the total pressure is modified circumferentially by superimposing an arbitrary Fourier series but keeping an uniform radial distribution.

The goal is to use three reduced-passages to capture the large-scale non-axisymmetric perturbations of the flow and predict the trigger of the stall process under distorted inflow, filtering the harmonics which are different than the nodal diameter at the inlet. The first are full-annulus simulations which are used as a high-order model for verification purposes. The second are PSM simulations with a single spatial harmonic or ND matching that of the inlet perturbation. This approach is very efficient and is used to map the design space. Using this technique either simple sinusoidal perturbations in the azimuthal direction or standard 60° or 90° inlet distortion

Figure 4.2.: Effect of inlet distortion amplitude at 100% of the design speed for $ND = 1$ computed with the PSM.

screens can be created.

4.2.1.1. Non-linear Stability

The effect of varying the amplitude of the inlet stagnation pressure first nodal diameter pattern on the fan stability at $\Omega_{rotor}/\Omega_{nominal}=100\%$ is investigated first with the present method. In this case the P_0 at the inlet can be expressed as $P_0 = \bar{P}_0 + \Delta P_0 \sin(M\theta)$, being M the harmonic index retained in the method. Solutions are obtained using the PSM just retaining the harmonic $M = ND$, as shown for instance in Fig. 4.1 for $ND = 3$. Figures 4.2 (a) and (b) show the stall margin as a function of the corrected mass flow. The blue circle with the minimum mass flow coincides with the last converged point with the single-passage steady version of the solver. At this OP the fan is unstable under infinitely small perturbations. As the mass flow is increased the fan is more tolerant to inlet perturbations. This process is sketched in Fig. 4.2. At the bottom the blue circles represent the mean operating point with the inlet perturbation included. This means that these points are the result of a time averaging procedure of the unsteady solutions. It can be noticed that the deviation with respect the speedline computed using single-passage

steady simulations is small. The stall margin loss can be defined as the deficit of stall margin with inlet distortion

$$SM_L = \frac{PR_{ref} - PR_s}{PR_s} \times 100\% \quad (4.3)$$

where PR_{ref} and PR_s are the total pressure ratio of the near stall point with clean flow and distorted inflow, respectively. It can be seen that as the amplitude of the 1st ND perturbation becomes higher, from an amplitude of 200 Pa in Fig. 4.2 (b) to 2000 Pa, the stall margin loss of the blade raises from 1.21% to 7.83%. The stall margin loss increases steadily with the distortion amplitude. This behaviour is usually explained by noting that for large total pressure perturbations the airfoils operate during part of the period in off-design conditions making some of the blades more prone to separation triggering the whole fan in stall.

Figure 4.2 (a) shows the non-linear stability map for the 1st ND computed with the PSM. This case exhibits a subcritical bifurcation, i.e.: OPs which are stable under small perturbations can become unstable in the presence of large disturbances. This instability process is non-linear and for small perturbations this phenomenon cannot be seen. Its simulation requires of fully non-linear analysis. Figures 4.2 (a) and (b) show that the farther apart the OP from the stall line the larger the disturbance required to drive the fan into a massive stall. Large perturbations can trigger an intrinsically non-linear stall process. This behaviour is well known and has been reported and simulated by several author's [80, 94, 96, 102].

Traditionally, fan stability has been experimentally and numerically investigated using inlet distortion screens to perturb the incoming flow. These perturbations are physically implemented partially blocking a sector of the intake. Here, the difference is that taking advantage of the PSM the contribution of the different NDs can be investigated independently preventing the interaction between them. The stability of the fan has been investigated in a nodal diameter basis retaining the spatial harmonic corresponding to the inlet perturbation solely.

Figure 4.3 depicts the non-linear stability map of the first four NDs at the inlet. The unstable regions are shaded in colour. It can be seen that the contribution of the different NDs to the non-linear stability is qualitatively different. The $ND = 1$ has a sub-critical bifurcation while that of the $ND = 3$ is supercritical. However, the stability of the NDs two and four is practically

Figure 4.3.: Stability conceptual map due to different amplitudes of inlet distortion distribution at 100% of the design speed and nodal diameters, ND , using the proposed PSM just retaining the harmonic $M = ND$.

not altered by the level of the perturbation. The four nearly straight lines depart from the same point, at least with the error of the method, which is the linear or small perturbation stability limit. The first ND has the largest sensitivity to inlet perturbations, and in practice, its behaviour is the one observed with realistic, non-sinusoidal, perturbations. The most surprising result of Fig. 4.3 is the stabilizing behaviour of $ND = 3$. The actual interpretation of Fig. 4.3 (c) is that the fan could operate beyond the linear stability limit if the sole forcing at the inlet corresponds to the ND three, and this is high enough. In this case the fan exhibits a periodic but stable response to this type of forcing. The combination of ND three with other ND s, such as the first, has not been further pursued.

The effect of the 1st ND inlet total pressure amplitude on fan stability has also been investigated at $\Omega_{rotor}/\Omega_{nominal} = 50\%$. It is concluded that, in this case, the stability of the fan depends very weakly on the perturbation amplitude for all the ND s and therefore the overall stability of the fan coincides with the obtained for the small perturbation regime. This means that the contribution of the inlet distortion to the change in the fan stall margin of different total pressure

Figure 4.4.: Stability conceptual map with clean inflow using PSM with a single harmonic M at 100% of the design speed.

patterns depends on the rotational speed [80].

4.2.1.2. Linear Stability

The stability of the fan-stage is now studied for clean inflow conditions. The simulations are set with a constant uniform total pressure at the inlet and a single harmonic, M , of the PSM is computed. This is physically equivalent to compute different sector sizes. For instance if $M = 3$ the problem would be equivalent to run a 120° sector and filter out all the contributions corresponding to the rest of NDs. The pressure ratio is modified by changing the level of the circumferentially uniform exit static pressure.

Figure 4.4 shows the linear stability threshold as a function of the spatial harmonic index or sector size. It can be seen that the most unstable ND is the third, but the difference between different NDs is not high. The zero nodal diameter is the most stable. This means that if single-passage simulations were run to drive the code to stall at constant speed, before this actually happened, the equivalent full-annulus simulation would transition to an unsteady rotating stall pattern. The points of Fig. 4.4 represent the crossing of the stability limit of the different NDs in Fig. 4.3 with the horizontal axis. The departing pressure ratio changes slightly with the ND, exactly as shown in Fig. 4.4.

This type of plot is difficult to construct using full-annulus simulations, especially if the stability limit of the different modes is close, since it is not possible to control the development of all

the unstable NDs when they are active simultaneously in a single simulation. According to Fig. 4.4, the fan cannot operate in a stable manner for pressure ratios higher than 1.65. However, Fig. 4.3 (c) suggests that if a high enough perturbation with the third nodal diameter pattern is introduced the fan could operate in a stable manner beyond this point, which is an interesting conclusion of this study due to its potential application.

Finally, it is important to highlight that while the most unstable nodal diameter in a linear sense is $ND = 3$, the most critical ND is the first, and the one which can be triggered by a finite-amplitude perturbation in the linearly stable region. This means that linear fan stability analysis is misleading in most of the practical situations.

4.2.2. Results

The goal of this section is to prove that though screen-induced distortion is complex since involves many Fourier spatial modes, the only relevant feature to characterise fan non-linear stability is the most non-linearly unstable nodal diameter. Although in the context of linear stability this statement is obvious, in the present problem which involves highly non-linear mechanisms this statement needs to be proven. To do that a numerical strategy has been developed to study the non-linear stability of a fan-stage under the effect of different total pressure distortion screens in the stationary frame of reference by approximating it using Fourier series and analysing the stability of each of the harmonics in a separate way.

A physically induced irregularity in the approaching flow can lead to a premature dynamic separation of the blades propagating along with the compressor circumferential direction in the sense of rotor rotation. This instability destroys the cyclic symmetry of the flow since typically a fraction of the wheel contains stalled airfoils while the remaining exhibits attached flow. The sinusoidal shape of the inlet distortion profile promotes the development of transient low-order spatial harmonics. This is in contrast to other types of inlet distortion where a stepped total pressure distortion screen is used to represent the blockage at the inlet [10, 12, 81]. However, as we will show in the following sections, both inlet distortion patterns can behave similarly.

Figure 4.5.: Full-annulus comparison between stable and unstable conditions with a first nodal diameter ($ND = 1$) perturbation with amplitudes of $\Delta P_0 = 1000 Pa$ and $\Delta P_0 = 3500 Pa$, marked as [a] and [b] in Fig. 4.3 (a) respectively, for the 1st harmonic component ($m = 1$) of the total pressure signal at 0.5 axial chords downstream of fan (see Fig. 3.9). $P_0 = 101325 Pa$.

The OP marked as point A ($\square$) in Fig. 4.2 (b) was chosen to investigate this point. The steady solution at 100% of the design speed with uniform inlet stagnation pressure at a near peak efficiency operating point is used as the initial solution for the unsteady simulations.

4.2.2.1. Stability Map Validation

The full-annulus solutions obtained for the points marked as [a] and [b] in Fig. 4.3 (a), and which correspond with two different perturbation levels of the 1st ND, are described next. Figure 4.5 (a) compares the the mass flow time evolution during the first 8 revolutions at the rotor outlet for two levels of perturbation, $\Delta P_0/P_0 = 1\%$ and 3.5% which correspond to stable and unstable full-annulus cases respectively. For the stable case, it can be seen how following the initial perturbation the mass flow converge to a stable solution, though the simulation is fully unsteady, whereas for the unstable case the mass flow drops by about 15% and has an erratic behaviour.

Figure 4.5 (b) displays the frequency spectrum in the relative frame of reference of the total pressure signal in a numerical probe located near the outer casing 0.5 axial chords downstream of the fan, see Fig. 3.9. The horizontal axis of Fig. 4.5 (b) represents the angular velocity of propagation of the spatial harmonic m relative to the rotor, $\Omega_m = \omega_m/m$, non-dimensionalised with the rotor speed, Ω_{rotor} . The spectrum of the stable case with $\Delta P_0/P_0 = 1\%$ shows a single perturbation propagating with the rotor speed caused by the total pressure distortion at

Figure 4.6.: Fourier approximation (left) and total pressure inlet distortion profile (right) for the 120° and 60° screens with 15% and 25%, respectively of inlet total pressure deficit, and the equivalent sinusoidal perturbation. ($P_{0,ref} = 101325 Pa$).

the inlet. The amplitude of this perturbation nearly coincides with that of the inlet meaning that its attenuation is very low. This case reduces to a pure forced response case with a single fundamental frequency.

On the contrary, the spectrum of the unstable case ($\Delta P_0/P_0 = 3.5\%$) is rich in frequencies due to the stall process. The fundamental frequency created by the inlet perturbation is also seen. Additionally, the propagation velocity of the stall cell which is about 25% of the rotor speed (in the relative frame of reference) is clearly distinguished. The unsteadiness created by the stall cell is nearly three times bigger than that created by the sinusoidal perturbation at the inlet. More detailed results of this unstable case will be discussed in the following lines.

4.2.2.2. Equivalent Inlet Distortion Principle

Complex inlet distortion patterns can be Fourier transformed and then the non-linear stability maps shown in Fig. 4.3 used to assess the fan stability in a nodal-diameter by nodal-diameter basis. As it has been mentioned before, it is unclear why this idea should work at all in a strongly non-linear problem when several NDs of finite amplitude are superimposed.

With the objective in mind of proving this hypothesis, two inlet distortions patterns with 120°

and 60° inlet screens were created and simulated using full-annulus analyses. The total pressure defect in the 120° and 60° inlet screens were chosen to be approximately 15% and 25% of the inlet total pressure respectively, leading in both cases to the same amplitude of the first harmonic, namely $\Delta P_0 \simeq 3500 Pa$. This case corresponds to point [a] in Fig. 4.3 (a) which according to the proposed methodology should be unstable. The two different virtual inlet screens and their Fourier content can be seen in Fig. 4.6. These simulations are compared with a third case consisting of a simple 1st ND sinusoidal pattern at the inlet with the same amplitude, $\Delta P_0 \simeq 3500 Pa$. It can be verified using Fig. 4.3, that the only unstable mode of the harmonic content of both screens is the 1st ND.

The hypothesis of the proposed approach has been confirmed using full-annulus simulations. From a stability point of view, the points [a] and [b] ($\Delta P_0 \simeq 1000 Pa$.) in Fig. 4.3 (a) computed using a 60°-sector, a 120°-sector and a sinusoidal perturbation lead to the same conclusion: Point [a] is unstable and Point [b] is stable. This is consistent with the location of both points with respect to the neutral line. Though the answer is correct, it could be argued that points [a] and [b] are located far away from the neutral line, and therefore this approach is only qualitatively correct.

To assess the accuracy of the stability criterion based on the analysis of the individual spatial Fourier modes, the non-linear stability threshold for the 60° inlet distortion screen has been found resorting to full-annulus simulations. Departing from point [a], the total pressure defect of the screen has been gradually decreased until approximately the stability limit obtained using the PSM which is $\Delta P_{0,c} \simeq 1800 Pa$, corresponding to a total pressure defect of the screen of approximately 15%. The inlet background total pressure is kept constant during the whole iterative process. An equivalent process was followed for point [b]. Points [c] and [d] are the results of this process, as shown in Fig. 4.3 (a), and bracket the stability limit for the amplitude of the 1st harmonic between $\Delta P_{0,c} \simeq 1800(1 \pm 0.1) Pa$. The iterative process based on a series of full-annulus simulations was stopped when the error in the perturbation was 10%, with respect to the threshold provided by the simplified PSM method. It can be stated that point [d] is stable whereas [c] is not. Further points, closer to the stability limit delivered by the PSM method,

Figure 4.7.: Comparison for the three different inlet distortion screens at 100% of the design speed. (a) and (d): sinusoidal screen, (b) and (e): 120°-screen, (c) and (f): 60°-screen. Top: Snapshot of the relative total pressure contours 0.5 axial chords downstream of the fan after 8 rotor revolutions non-dimensionalised with the inlet total pressure amplitude of the 1st spatial harmonic, ΔP_0 . Bottom: non-dimensional relative angular phase velocity of the spatial harmonics, Ω_m/Ω_{rotor} , for the three inlet distortion screens of a total pressure probe near the outer casing 0.5 axial chords downstream of the fan (see Fig. 3.9) non-dimensionalised with ΔP_0 .

were not attempted.

4.2.2.3. Statistically Converged Flow Description

Figures 4.7 (a), (b) and (c) show the instantaneous contours of the relative total pressure 0.5 axial chords downstream of the fan after 8 rotor revolutions for the point [a] in Fig. 4.3 (a), where $\Delta P_0 = 3,500\text{Pa}$ and $\text{PR}=1.56$. When the flow is destabilized a stall cell is created in a low total pressure region near the tip of the blade created by the inlet distortion. The separation begins to grow in this low-pressure region and the stall is triggered. Once the perturbation created in the inlet impinges the rotor, two more revolutions are needed to form a fully developed stall cell. It was shown, both experimentally [102] and numerically [12], that the flow disturbances with frequencies different from the fundamental one associated with the inlet perturbation trigger

the stalling of the blade. Such frequencies cannot be modelled with single-passage method using phase-shifted boundary conditions since the basic assumption is that there is a unique and known fundamental frequency. This is the underlying reason why full-annulus solutions have been used for these complex cases until now. However, the PSM, even with a single spatial harmonic, can reproduce the frequency spectrum quite decently with the corresponding reduction in computational resources. It has been previously demonstrated in Chapter 3 that the present method can reproduce fan transients caused by inlet distortion with a low number of spatial harmonics.

To further study this statistically converged quasi-periodic state the time-varying amplitude of the spatial harmonics has been Fourier transformed in time. First, the spatial Fourier transform of the selected points is performed and then the Fourier coefficients are Fourier transformed in time again. The tracking point where the time-varying total pressure signal is studied, is located in the rotor frame of reference near the casing 0.5 axial chords downstream of the fan trailing-edge. It was found that the flow takes about 8 rotor revolutions to reach a converged quasi-periodic state. To obtain a clean signal the initial part of the transient was discarded in the analysis. The dimensionless phase velocity spectra are presented in Figs. 4.7 (d), (e) and (f) where the horizontal axis represents the angular velocity of propagation of the harmonic relative to the rotor, $\Omega_m = \omega_m/m$, non-dimensionalised with the rotor speed, Ω_{rotor} . It can be seen that the spectrum is rich in frequencies as a consequence of the interaction of the stalled cells with inlet non-uniformity.

Synchronous Response The synchronous response of all the fan harmonics due to the inlet non-uniformity travels with the rotor velocity in the relative frame of reference and can be distinguished in Figs. 4.7 (d), (e) and (f) at $\Omega_m/\Omega_{rotor} = 1$. It is important to recall at this point that Fig. 4.7 (bottom line) displays the spatial mode amplitude as a function of the phase speed $\Omega_m = \omega/m$. and therefore all the incoming are superimposed at $\Omega_m/\Omega_{rotor} = 1$. The amplitude of the first harmonic, $m = 1$, is the same for the three cases and approximately equal to $\Delta P_0 \simeq 3500 Pa$. Figures 4.7 (e) and (f) which correspond to the 120° and 60° screens respectively, show contributions as well from higher order modes. The mode $m = 2$ appears both in the 120°

and 60° cases. However, this is not the case for the mode $m = 3$, whose contribution is zero for the 120° case. This is consistent with the frequency content of the incoming perturbation, see Figs. 4.6 (c) and (d). It is important to note the third harmonic of the 120° -screen is zero. This is why it does not show up at $\Omega_m/\Omega_{rotor} = 1$ in Fig. 4.7 (e). Though this conclusion is obvious, it is important to remind it for a correct interpretation of Fig. 4.7.

Asynchronous Response The patterns travelling at a speed different from the angular rotor velocity appearing in Figs. 4.7 (d), (e) and (f), correspond to that generated by the stall process. When the inlet perturbation impinges the rotor, low frequencies whose main spatial content corresponds to the 1st ND ($m = 1$) consistently appear, see Figs. 4.7 (d), (e) and (f). The propagation velocity of this pattern is about 25% of the rotor velocity in the relative frame of reference in all the three cases. A single stall region is created in all three cases. It is important to highlight that the 1st ND diameter was identified by the non-linear stability analysis as the most promising candidate to be triggered by a finite amplitude excitation.

The tip leakage vortex shows a periodic behaviour as the rotor enters and exits the distorted region. Rotating stall will happen when the flow separation generated in the distorted region is not removed in the clean flow area [12]. These disturbances are almost constant along time, reaching a quasi-steady state. After about 3 rotor revolutions (see Fig. 4.5), the mass flow decreases to stall levels and the low-frequency content of the spectrum increases, what is a natural consequence of stall process.

The first ND has the largest contribution to the stall cell and its unsteadiness level is three times bigger than that created by the inlet non-uniformity. The spectrum has a complex harmonic content but in overall they are alike. Obviously, the spatial content of the simulation performed with the PSM method using a single spatial harmonic has fewer spatial scales but the frequency content is similar. The method can reproduce the one stall rotating cell of the full-annulus solution with reasonable accuracy just retaining the most unstable nodal diameter. A detailed discussion of this solution is outside of the scope of this paper.

The purpose of this exercise is to prove that the proposed method provides a similar solution as

the full-annulus case, both in terms of spatial and temporal harmonics in the final statistically converged state. In this operating point and with the selected level of forcing the solution is unstable but with a limited amount of unsteadiness and a relatively simple spectrum. The behaviour of the different cases, with different inlet distortion screens, but with the same amplitude ($\Delta P_0 \simeq 3500 Pa$) of the most non-linearly unstable nodal diameter are similar. It is concluded that the predictions derived from the non-linear stability maps are valid. The three selected cases are unstable as it was predicted.

It is interesting to compare the solution of the final saturated state obtained in this case against the results presented in Chapter 3. In this section it is concluded that the 1st nodal diameter of the distorted inflow was the most nonlinearly unstable. Moreover, when the inlet perturbation of the 1st nodal diameter was $\Delta P_0 = 3.5 kPa$ it turned out that the 1st harmonic was the most unstable in the final saturated state as well. However, in the case studied in Fig. 3.18 and Fig. 3.19, for a total pressure perturbation of 1st nodal diameter distortion at the inlet of $\Delta P_0 = 5 kPa$, the 2nd harmonic has the largest contribution to the final flow solution. It can be concluded by comparing both results that, by increasing the amplitude of the total pressure distortion at the inlet, the non-linear contribution of the different harmonics to the final state changes. Besides, the most unstable or dominant spatial harmonic in the final state is not necessarily the same as that of the most unstable perturbation at the inlet. By comparing also the high total pressure amplitude distortion case with $\Delta P_0 = 10 kPa$ (see Fig. 3.22 and Fig. 3.23), where the 3rd harmonic had the largest contribution to the final state solution, against the results reported in this section, it can be concluded that by increasing the amplitude of the inlet disturbance the non-linear contribution of the higher harmonics to the final saturated state is increased as well.

4.2.2.4. Rotating stall Removing Inlet Distortion Forcing

This section aims to assess the question regarded by the effect of the distorted inflow on the stall process, once the solution becomes unstable. In order to do that, as it has been arised before, the solution reaches a quasi-periodic state after 8 rotor revolutions, triggering a stall process where an associated frequency can be reproduced. By this instant, the distorted pattern of $ND = 1$

Figure 4.8.: Comparison between distorted and clean inflow after 8 rotor revolutions, once the solution reaches a quasi-steady state stall conditions with a first nodal diameter ($ND = 1$) perturbation with amplitude $\Delta P_0 = 3500 Pa$, marked as [a] in Fig. 4.3(a), for the 1st harmonic component ($m = 1$) of the total pressure signal at 0.5 axial chords downstream of fan (see Fig. 3.9).

is removed and simulation are re-launched imposing a uniform total pressure at the inlet. The evolution of two simulations are then compared; one solution carries on the evolution from 8 rotor solutions with a distorted inflow of $\Delta P_0 / P_0 = 3.5\%$, point marked as [a] in Fig. 4.3 (a), and the other solution is computed after 8 rotor revolutions, when the stall has reached a quasi-steady state with a clean inflow imposed at the inlet.

Figure 4.8 displays the frequency spectrum in the relative frame of reference of the total pressure signal in a numerical probe located near the outer casing 0.5 axial chords downstream of the fan, see Fig. 3.9. The horizontal axis of Fig. 4.8 represents the angular velocity of propagation of the spatial harmonic m relative to the rotor, $\Omega_m = \omega_m / m$, non-dimensionalised with the rotor speed, Ω_{rotor} .

The spectrum of the distorted inflow conditions is almost the same that the studied in Fig. 4.7 (a), since we simulate 10 rotor revolutions more after the last iteration in that section. The fundamental frequency created by the inlet perturbation is clearly seen, as well as the propagation velocity of the stall cell which is about 25% of the rotor speed (in the relative frame of reference) is clearly distinguished. This spectrum is rich in frequencies, different than the mentioned above. On the contrary, clean inflow simulation reveals a cleaner frequency spectrum, since the

contribution of the distorted inflow has been removed.

First conclusion is that, for a converged quasi-periodic state under stall conditions, the instability associated to low velocities and pressure patterns due to stall conditions is invariant to the non-uniformity of the inlet pattern in the circumferential coordinate once stall conditions are triggered.

Second conclusion is that these frequencies different than the stall cell propagation velocity (about 25% of the rotor speed in the rotor frame of reference) are caused by the inlet non-uniformity in the distorted perturbation of 1ND. In the clean inflow simulation, where the inlet is not perturbed, the only frequency in the spectrum of Fig. 4.8 is the one related with the stall cell speed.

4.3. Outlet Static Pressure Perturbation Stability Analysis

Turbomachinery blades are susceptible to non-uniform flows generated by inlet distortions, wakes and upstream and downstream pressure disturbances from adjacent blade rows. Pressure non-uniformity is particularly strong in the fan bypass duct due to the existence of pylons and struts which support the fan casing. The potential field disturbances of these obstacles in the form of a circumferentially varying pressure can result in forced excitation of the fan rotor, noise generation and increased aerodynamic losses. These pressure waves can eventually propagate upstream through the fan, and as a result, rotor blades are subject to unsteady fluctuations causing the fan to operate with a non-uniform exit pressure. Operation in a distorted flow-field also affects fan performances.

Analytical models were used for the rotor and stator rows by some researchers. [110] used a two-dimensional compressible semi-actuator disk model based on the theory of small perturbations for a fan stage. Static pressure disturbances due to downstream pylons or struts were separately obtained and imposed at the exit of the fan stage model. However, time-accurate

CFD simulations are frequently used in design-optimization systems and are becoming more and more widespread within the industry [111, 112]. Since the mechanism of rotor interaction with a downstream potential pressure field is inherently unsteady, URANS analyses of a 360° full computational model of the whole stage with downstream obstructions is required to correctly resolve the problem. [113] performed three-dimensional unsteady viscous analyses for two fan rotor-stator-pylon configurations with different axial gaps between the stator and the pylon, and compared their results with experimental data. The common characteristic of these numerical strategies based on URANS simulations is their high computational cost. Modeling the aerodynamic effects using full-annulus transient simulations though feasible [114, 115], is still too expensive for routinely used in design offices.

Passage-Spectral Method (PSM) is proposed in this study to reduce the computational cost of these unsteady and non-axisymmetric simulations and simplify the study of the fan rotor stability under the effect of a distorted exit static pressure. Despite the number of struts or pylons and their geometrical complexity, any outlet static pressure distortion pattern can be Fourier decomposed. The stability of the spatial harmonics content can be assessed separately using the proposed PSM filtering the contribution of the rest of the circumferential modes of the downstream perturbation. The validity of this approach has been shown previously for inlet total pressure distortions. In this model, just three reduced-passages are retained. As mentioned before, the proposed method assumes that the circumferential wavelength of the potential disturbances is much larger than the blade spacing. In this section first, the non-linear stability of each nodal diameter at the outlet is investigated as a function of the perturbation amplitude taking advantage of the use of the novel method. Next, full-annulus simulations are carried out to increase the credibility of the stability maps simulating the potential effect of two pylons and its equivalent stability study just retaining the most unstable nodal diameter.

4.3.1. Fan-Stage Stability

This section aims at studying the non-linear aerodynamic stability of a generic fan blade operating at a constant speed under the potential effect caused by a circumferential distribution of

static pressure at the outlet, which is parametrized in terms of its nodal diameter and static pressure amplitude. The performances and stability of axial flow fans and compressors is affected by the circumferential variations in exit static pressure produced by thick struts or other components if they are not located far enough from the fan rotor. These flow disturbances are usually dominated by long circumferential wave-lengths (i.e. much longer than the blade pitch) which makes the Passage-Spectral Method well suited for their analysis since it avoids the use of full-annulus simulations. Moreover, the method allows investigating the independent contribution of the different NDs efficiently. It is important to point out that this is not the case in non-linear full-annulus simulations since when the perturbations are large, and the linear stability regime is by-passed, the non-linear interaction amongst the different NDs is unavoidable.

The potential field of the pylons also causes performance and integrity issues in the fan blades. Because of the detrimental effect of the pylons potential fields, a large amount of work has been devoted over the years to controlling them. The location and size of the pylon is very much a matter of overall engine and installation architecture and it represents a datum for the design of the flow-path. In this regard, the method has been adapted to simulate an arbitrary single ND perturbation, as sketched in Fig. 4.1, simulating the equivalent effect of different number of pylons downstream, using solely three reduced passages to fit the desired wave-length over a sector of $2\pi/ND$ degrees, filtering the non-linear contribution of the rest.

The NASA rotor 67 has been chosen as the test vehicle as in the previous section. Design parameters and details of the mesh grid (see Fig. 3.1 and Fig. 3.2) have already been described and outlined in previous section in Tab. 3.1 at %100 of the design speed. The selected operating condition in the fan map for the steady simulation at 100% of the design speed with uniform static pressure at the outlet is represented in Fig. 3.4 as point A ($\square$) at a near peak efficiency operating point, which is used as the initial solution for the unsteady simulations. Unsteady simulations have been conducted using 400 time steps per rotor revolution. The number of pseudo time steps per physical time step was set to 60, which is large enough to achieve the convergence criterion of the inner iteration of the dual time step. The base flow is uniform and axial at the outlet, but the static pressure is modified circumferentially by superimposing an

arbitrary Fourier series but keeping the radial distribution of the static pressure.

Two types of simulations are used in this work. The first are full-annulus simulations which are used as a high-order model for verification purposes. The second are PSM simulations with a single spatial harmonic or ND matching that of the outlet static pressure perturbation. This approach is very efficient and is used to map the design space following the same strategy than in the previous section due to inlet distortion effect on the fan-stage stability.

4.3.1.1. Non-linear Stability

There have been various suggestions to prevent interaction on the fan rotor blades due to pylon potential field which include: the introduction of additional vanes to divert the flow around the pylon and increase the axial distance between the fan and the pylon. Since the pylon induced field has a low axial decay rate, the option of increasing the axial gap between the fan and the pylon becomes unacceptable as it will require a considerable increase in engine size. Both modifications are unattractive as they will almost certainly result in increased size and weight of the engine and result in the introduction of additional components. Modifying the pylon shape has also been considered by various researchers [116, 117, 118]. The aim of this section is to study the influence of different circumferential modes of the static pressure pattern downstream on the fan in order to bring some light to the influence of modifying the pylon shape in a preliminary design process of the non-linear stability of the fan-stage.

Traditionally, fan stability has been experimentally and numerically investigated using full model including struts in the computational domain as well as full-annulus geometries to study the potential effect of the distorted static pressure perturbation downstream of the fan rotor blades. These complex static pressure distribution in the circumferential coordinate induced by the presence of struts or pylons can be Fourier decomposed in spatial harmonics. Here, an approximation has been made to take into account only one of these spatial harmonics, taking advantage of the PSM where the contribution of the different NDs can be investigated independently preventing the interaction between them. The basic assumption is that the scale of the relevant disturbances produced by static pressure distortion at the outlet are much larger

than the blade pitch, and then a few low-order spatial harmonics suffice to predict flow behaviour. The stability of the fan is then investigated in a nodal diameter basis retaining the spatial harmonic corresponding to the outlet static pressure perturbation solely. The pressure ratio is modified by changing the mean level of the circumferentially average exit static pressure.

The effect of circumferentially varying the amplitude of the outlet static pressure for the first four nodal diameter patterns on the fan stability is investigated with the Passage-Spectral Method at $\Omega_{rotor}/\Omega_{nominal}=100\%$. In this case, the static pressure at the outlet can be expressed as $p_s(r, \theta) = \bar{p}_s(r) + \Delta p_s \sin(M\theta)$, being M the harmonic index retained in the method. Solutions are obtained using the PSM just retaining the harmonic $M = ND$. It is noteworthy that these perturbations have a travelling-wave form in the rotor frame of reference, and for the frequencies and nodal diameters studied in this work. These waves are cut-on and develop upstream without attenuation, at least for the frequencies and nodal diameters studied in this work.

The study of the amplitude effect of a static pressure perturbation downstream on the fan has been carried out taking into account realistic circumferential patterns previously reported in the literature. Enoki et al. [113] conducted aerodynamic and acoustic tests for a transonic fan stage with a simulated top pylon. The potential pressure field that was created by the engine pylon was about 15% of the mean static pressure three chords downstream from the fan. Taking into account the actual static pressure at the outlet of the simulations carried out at the aerodynamic design point, the amplitude of the static pressure perturbation at the exit has been varied between $\Delta p_s = 0$ and 20.000 Pa for different nodal diameters.

The non-linear stability maps for the first four NDs computed with the PSM are shown in Fig. 4.9. The unstable regions are shaded in colour. The blue circle with the minimum mass flow coincides with the last converged point with the single-passage steady version of the solver. At this OP the fan is unstable under infinitely small perturbations. As the mass flow is increased the fan is more tolerant of outlet perturbations. The stability of the first and third NDs, see Figs. 4.9 (a) and (c), are practically not altered by the perturbation level. These cases exhibit nearly straight vertical lines departing from the same point, at least within the error of the method, which is the linear or small perturbation stability limit. This instability process is non-linear,

Figure 4.9.: Stability conceptual map due to different amplitudes of static pressure patterns at the outlet and nodal diameters using the PSM just retaining the harmonic $M = ND$ at 100% of the design speed.

since the perturbations have finite amplitude, and for small perturbations, this phenomenon cannot be seen. Its simulation requires of fully non-linear analyses.

It can be seen that the contribution of the different NDs to non-linear stability is qualitatively different. The most surprising result of Fig. 4.9 is the stabilizing behaviour of $ND = 2$ and 4. The stability boundary of these nodal diameters exhibits a supercritical bifurcation, i.e. OPs which are unstable under small perturbations at the outlet, $\Delta p_s/p_s \ll 1$, can become stable in the presence of large disturbances at the outlet. The actual interpretation of Figs. 4.9 (b) and (d) is that the fan could operate beyond the linear stability limit if the sole potential effect at the outlet would correspond to the $ND = 2$ or 4, and the amplitude of the static pressure disturbance at the exit is high enough. In this case, the fan exhibits a periodic but stable response to this type of potential effect downstream on the fan. The combination of these stable NDs with other NDs, such as the first, has not been further pursued. Comparing the results for the first four NDs in Fig. 4.9 can be concluded that the first and the third NDs have the largest sensitivity to outlet static pressure perturbations, and in practice, their behaviour is the one observed with realistic,

non-sinusoidal, perturbations.

The stability of the fan-stage under small perturbations is studied next. The simulations are set with a constant circumferentially uniform static pressure at the exit ($\Delta p_s = 0$ in Fig. 4.9) and a single harmonic, M , of the PSM. This is equivalent to compute different sector sizes or in the insets of Fig. 4.9. It can be seen that the most unstable ND is the third, but the difference between different NDs is very small. It is concluded that the critical pressure ratio derived from the small perturbation analysis has a weak dependence with the ND and the departing point of all the non-linear stability limit for all the NDs is essentially the same.

This type of plot is difficult to construct using full-annulus simulations, especially if the stability limit of the different modes is similar in the parameter space since it is difficult to control and track the development of the different unstable NDs when they are active simultaneously in a single simulation. However, Fig. 4.9 suggests that if high enough perturbations with $ND = 2$, and 4 are introduced the fan could operate in a stable manner beyond this point, which is an interesting conclusion of this study due to its potential application. Finally, it is important to highlight that, the most critical ND is the first and the one which can be triggered by a finite-amplitude perturbation in the linearly stable region.

It is illustrative to compare these results with the more classical problem of intake distortion where a total pressure defect is set at the inlet. This problem has been addressed in the previous section studying the effect of the inlet distortion on the NASA rotor 67 fan stability following the same approach. In that case, the $ND = 1$ total pressure perturbation at the inlet has a sub-critical bifurcation while here the sensitivity of the fan to the amplitude of a potential perturbation at the exist is very small. The most surprising result for the case of intake distortion is the strongly stabilizing behaviour of a $ND = 3$ total pressure distortion pattern at the inlet, as shown in Fig. 4.3 (c). This stabilizing effect can be shown also here for a potential perturbations at the exit with $ND = 2$ and 4 patterns, see Figs. 4.9 (b) and (c), but this effect is much stronger in the case of intake distortion. It can be concluded that the fan stability sensitivity to finite amplitude periodic perturbation is much higher for intake distortion case than for potential perturbations associated to struct or pylons at the exit.

Figure 4.10.: Fourier approximation (left) and static pressure outlet profile (right) for two 7°-sector static pressure perturbation with 30% of outlet static pressure deficit and the equivalent sinusoidal $ND = 2$ potential effect. ($p_{s,ref} = 120000 Pa$).

4.3.2. Results

In aircraft gas turbine engines, there are normally two pylons located in the upper and lower parts of the bypass duct [119]. They are utilized not only as structural elements, but also as fairings for the passage of oil lines and electrical cables. The pylons are usually of sufficient size to produce a significant potential flow disturbance. As previously concluded in the last section for the non-linear stability of screen-induced distorted inflow, the goal now is to prove that though the complexity of potential perturbation downstream the fan-stage, since involves many Fourier spatial modes, the only relevant feature to characterise fan non-linear stability is the most non-linearly unstable nodal diameter. This statement needs to be proven due to the presence of highly non-linear mechanisms in potential effect of downstream pylons or struct. To do that a numerical strategy has been developed to study the non-linear stability of a fan-stage under the effect of a static pressure distortion screens in the stationary frame of reference by approximating it using Fourier series and analysing the stability of each of the harmonics in a separate way. With the objective in mind of proving this hypothesis, the potential perturbation associated to the presence of two pylons is approximated by simulating two 7°-sector static pressure perturbation at the outlet using full-annulus analyses.

The OP marked as point A (□) in Fig. 3.4, near the peak efficiency, was chosen to investigate this point. Steady solutions at 100% of the design speed with uniform outlet static pressure are used as the initial solution for the unsteady simulations.

4.3.2.1. Equivalent Outlet Potential Effect Principle

The presence of two pylons is approximated by two 7°-sector static pressure perturbation downstream of the fan. The static pressure virtual effect due to the presence of two pylons of 7°-sector is decomposed in their Fourier content in Fig. 4.10. The aim is to study the effect on the stability of the fan by including all the harmonics in Fig. 4.10 (a), and compare the stability threshold by using the non-linear stability maps shown in Fig. 4.9, which includes only the effect of an isolated nodal diameter. The static pressure defect in the 7°-sector pylon distortion pattern were chosen to be approximately 30% of the mean static pressure at the outlet, as shown in Fig. 4.10 (b), leading to an amplitude of $\Delta p_s \simeq 2500 \text{ Pa}$ for the 2nd harmonic component, as shown in Fig. 4.10 (a). This static pressure defect of 30% was chosen according to similar results by other researchers [113].

The reason to select two pylons is to study the stability effect of an even number nodal diameters which the flow exhibits a supercritical bifurcation and have a stabilizing effect, as shown in Figs. 4.9(b) and (d), and verify that the only unstable mode of the harmonic content contribution by using all the harmonics in Fig. 4.10 (a) is the 2nd nodal diameter.

To assess the accuracy of the stability criterion based on the analysis of the individual spatial Fourier modes, the non-linear stability threshold for the two 7°-sector perturbation due to the presence of virtual pylons has been found resorting to full-annulus simulations. Full-annulus simulations have been conducted aiming not only at validating the results obtained in the previous section but also to show the transient behaviour when the flow-field becomes unstable. The full-annulus solutions for the two 7°-sector perturbation obtained for the points marked as [a] and [b] in Fig. 4.9 (b) are discussed next. Departing from point [a], the operating total pressure ratio has been gradually increased until approximately the stability limit obtained using the PSM in Fig. 4.9 (b). Points [a] and [b] are the results of this process, as shown in Fig. 4.9 (b), and

Figure 4.11.: Full-annulus comparison between stable and unstable conditions with two 7°-sector static pressure perturbation at the outlet for two total pressure ratio of $PR = 1.64$ and $PR = 1.65$ marked as [a] and [b] in Fig. 4.9 (a), respectively. Mass flow comparison (left) and pressure signal (right) at 0.5 axial chords downstream of fan near the casing (see Fig. 3.9) for the 2nd harmonic component ($m = 2$), non-dimensionalised with the distortion amplitude at the outlet, $\Delta p_s = 2500 \text{ Pa}$.

bracket the stability limit for the amplitude of the 2nd harmonic between $PR \simeq 1.64(1 \pm 0.01)$.

Figure 4.11(a) compares the mass flow time evolution during the first 20 revolutions at the rotor inlet for the operating points [a] and [b], corresponding to pressure ratios of $PR = 1.64$ and 1.65, respectively. For the stable case (operating point [a]), it can be seen how following the initial perturbation the mass flow converges to a stable periodic solution, though the simulation is fully unsteady, whereas for the unstable case (operating point [b]) the mass flow drops by about 50% after approximately 12 rotor revolutions and has a chaotic behaviour, see Fig. 4.11 (a) for the $PR = 1.65$ case.

Figure 4.11 (b) displays the time evolution of the second circumferential harmonic ($m = 2$) of the static pressure during the first 20 revolutions in a numerical probe located near the outer casing 0.5 axial chords downstream of the fan, see Fig. 3.9. The evolution of both signals in Fig. 4.11 (b) corresponds to the stable and unstable cases at the two different operating points, [a] and [b], previously mentioned. As can be seen in Fig. 4.11 (b) the effect of imposing a rich spatial harmonic content due to the presence of two 7°-sector perturbation at the outlet, described in Fig. 4.10 (a), is seen by the rotor as an upstream travelling-wave and sensed in the numerical probe. The stable case gives rise to a periodic disturbance in the flow as it could be expected. The amplitude of this perturbation is smaller than the imposed amplitude of the

static pressure perturbation at the outlet ($\Delta p_s = 2500 Pa$ for the 2^{nd} harmonics content) meaning that the pressure wave travelling upstream has been attenuated. This is in agreement with the results and conclusions derived by other researchers using either numerical and experimental data [113] or an actuator disk model [110].

After approximately 15 rotor revolutions, the second harmonic of the static pressure at the numerical probe located downstream of the fan blade is triggered and experiences rapid exponential-like growth, as shown in Fig. 4.11 (b), which indicates a rapid development of the non-axisymmetric flow in this region. Once the stall process is triggered, the second harmonic ($m = 2$) is shown to be persistent in time during the next 5 rotor revolutions. Here only the time evolution of the second harmonic is shown since is the most unstable one and in consequence the most relevant for the stall breakdown of the fan-stage. At this unstable operating conditions, $PR = 1.65$, the circumferential static pressure distortion with $ND = 2$ and amplitude of $\Delta p_s = 2500 Pa$ triggers an unstable behaviour of the fan blade that further develops into a fully stalled flow.

From a stability point of view, the points [a] and [b] ($\Delta p_s \simeq 2500 Pa$) in Fig. 4.9 (b) computed using two 7° -sector perturbation lead to the same conclusion: Point [a] is stable and Point [b] is unstable. This is consistent with the location of both points with respect to the neutral line. These two OPs have a slightly different pressure ratio, $PR=1.64$ and 1.65 , and a 2^{nd} ND potential perturbation at the exit with an amplitude of $\Delta p_s = 2500 Pa$. The hypothesis of the proposed approach has been confirmed then using full-annulus simulations. The stability threshold derived using PSM simulations has been verified using full-annulus simulations and a complex outlet static pressure pattern at these operating conditions. This is in good agreement with the predictions of the PSM including only the effect of the second nodal diameter, $ND = 2$. The bracket of the error in terms of total pressure with respect the full-annulus simulation for two 7° -sector pattern is below 1%, as shown in Fig. 4.9 (b). Further points, closer to the stability limit delivered by the PSM method, were not attempted.

4.4. Proposed Methodology

This section aims at proposing an efficient strategy that can be used to assess fan stability subject to inlet and outlet distortion quickly.

Firstly, for the selected fan blade and OP, the non-linear stability maps equivalent to those shown in Fig. 4.3 and Fig. 4.9 should be computed. On the view of the shape of the stability boundary for all the NDs two points per nodal diameter should suffice. If just the four first NDs are computed this analysis could be conducted overnight in a modest cluster with 16 GTX 1080ti GPUs.

Secondly, the inlet and outlet distortion patterns of interest is Fourier decomposed and the stability of its first spatial harmonics assessed using the previously computed non-linear stability maps. This process is inexpensive and the stability of a variety of these inlet and outlet perturbations could be readily assessed.

Finally the suspicious patterns could be assessed using either a full-annulus analysis or a PSM simulation retaining more spatial harmonics.

The impact of the different airfoil shapes or loadings could be quickly assessed also if some assumptions are made about the relevant NDs. If it is concluded that only the most unstable ND is relevant just the PSM simulation of this ND need to be kept in the loop.

The speed-up factor, R , of the PSM with respect a full-annulus simulation is directly related to the ratio between the number of reduced-passages samples used to approximate the simulation, and the number of blades, which in this case is 22, since the overhead of the method is negligible. The speed-up factor R , as outlined in Eqn. 2.13, for the new methodology with respect the direct full-annulus solution using three reduced-passages samples in this case is then $R = 22/3 = 7$. Taking into account that the same stall process and stability behaviour can be simulated retaining just the most unstable ND a significant speed-up with respect the use of realistic more complex screens can be obtained. A single full-annulus solution can be converged overnight in a cluster of 16 commodity GPUs assuming that the case is run for about 10 rotor revolutions. The non-linear stability maps can be computed in about the same time in the same

cluster.

4.5. Conclusions

Taking advantage of the Passage-Spectral Method, a simplified efficient methodology to assess the effect of intake distortion on fan stability is proposed. Unsteady simulations with various inlet distortion patterns have been carried out for the NASA rotor 67 fan blade to investigate its aerodynamic stability under the effect of different distortion screens. The reduced order Passage-Spectral Method and full-annulus representations of the fan have been used. The PSM has been slightly modified aiming at handling inlet distortion screens in a nodal diameter by nodal diameter basis efficiently without making any hypothesis about its temporal periodicity.

The proposed methodology is based on the heuristic assumption that the spatial harmonics of the inlet and outlet pressure perturbations can be Fourier decomposed and evolve independently, even in the non-linear stability regime. However, the method makes no hypothesis about the interaction between the different spatial modes. In this sense the method is fully non-linear and independent non-linear stability maps for each nodal diameter can be constructed. Stall point for one of the distorted screen pattern has been found with full-annulus simulations to estimate the error, which is approximately 10%, in the iteration procedure to build the stability map and verify the stability analysis with a full-annulus solution. The harmonic content of the three final quasi-periodic states have been characterised concluding that all the three are very similar since they are dominated by the non-linear instability of the 1st nodal diameter. It is concluded that at the design operating point the most non-linearly unstable nodal diameter is the first which exhibits a sub-critical bifurcation. Another conclusion is that the instability associated to low velocities and pressure patterns due to stall conditions is invariant to the non-uniformity of the inlet pattern in the circumferential coordinate. Once the solution has been converged to a quasi-steady state under stall conditions, the forcing due to a non-uniformity in the inlet does not affect to the stability of the system.

On the other hand for outlet static pressure distortions it is concluded that the fan stability

dependence with the amplitude of the perturbation is weaker than in the case of intake distortion. For perturbations with an odd number of nodal diameters, the non-linear stability analysis leads to the same conclusions as to the small amplitude linear stability analysis. However, if the perturbations have an even number nodal diameters, the flow exhibits a supercritical bifurcation and have a stabilizing effect. The presence of second and fourth nodal diameters has a non-linear stabilizing effect on the fan rotor blades. The stability threshold for the 2nd nodal diameter has been verified by approximating the effect of two pylons using full-annulus simulation. The error has been estimated at about 1% in terms of total pressure ratio. This conclusion has been derived using an iterative procedure involving full-annulus solutions.

The proposed method can predict the non-linear effect produced in a complex full-annulus simulation and capture the stability limit in a efficient way. The efficiency of the methodology is based on two cornerstones: the analysis of only the most non-linearly unstable nodal diameter and the novel Passage-Spectral Method which does not require the use of full-annulus simulations. In the author's opinion, this is the first time that clear conclusions are drawn on the influence of inlet and outlet distortions shape on fan stability.

5. Unsteady Flows in Rotating Cavities

Contents

5.1. Results for Rotating Cavities	128
5.2. Conclusions	154

Rotating instabilities in different cavity configurations are studied in this chapter as validation cases to the application of the proposed method, including several types of rotating disk systems, such as shrouded rotor-rotor cavities in compressors or rotor-stator cavities in turbines. In these two-disk flow, boundary layers develop at high rotation velocities caused by a differential rotation rate between solid boundaries and the fluid in rigid-body rotation in the cavity core, where unsteady and non-axisymmetric instabilities at the inviscid core region appear under certain conditions. These are associated with large-scale unsteady flow structures containing frequencies unrelated to the main annulus blading. These patterns have been identified in many experimental and numerical studies. These unsteady large-scale flow structures of the order of the rotor circumference have been shown in some cases to reduce sealing effectiveness and are difficult to predict with conventional steady or unsteady Reynolds-Averaged Navier-Stokes (RANS) simulations using a sector of the full wheel. Improved understanding of the underlying flow mechanisms of these phenomena, and how these could be controlled is needed to improve the sealing efficiency and robustness of gas turbines. The identification and characterization of these purely unsteady mechanisms should improve prediction methods and lead to new, more

Figure 5.1.: Sketch of disk cavities in a gas turbine [2].

efficient control strategies for these relevant flows for the turbomachinery community.

A briefly description about the baseline flow in rotating cavities is given in this chapter, as well as full-annulus comparison with the ultimate goal to show a different application of the proposed method into this complex non-axisymmetric and unsteady cases in turbomachinery. The proposed Passage-Spectral Method is applied in different cavity configurations to show the potential of the methodology in predicting these rotating instabilities where the fundamental frequencies are not known beforehand, and validate the method against full-annulus geometries in three configurations: a rotor-stator cavity, a rotor-rotor compressor cavity and a rim seal cavity. The goal of this chapter is to the proposed method to identify the spatial harmonics in the cavity cores to demonstrate the application of the method to a different configuration.

5.1. Results for Rotating Cavities

After several decades of research, a considerable body of knowledge on rotating flows in cavities is available from different geometries and simplified test rigs. However, comparisons of CFD predictions with measurements have often shown disparities, limiting the confidence in design predictions and motivating more detailed experimental and computational research. CFD and

experiments have established the intrinsically unsteady nature of many rotating large-scale unsteady flow patterns in the inviscid core of rotating cavities. The importance of the main-flow, and particularly the associated circumferential pressure variations was confirmed in early experimental studies. The first published experimental observation of these large-scale unsteady flow modes was reported by Cao et al. [22]. Since then several experimental and numerical investigations have reported evidence of large-scale unsteady flow patterns in different cavity geometries [25, 28, 33, 120, 121, 122]. URANS simulations show also rotating large scale flow structures unrelated to the rotating blades in various configurations, including examples with and without a forcing main annulus flow [123]. The occurrence of such flow structures has been confirmed by experiments, although when compared to URANS models these tend to show less distinct features and frequencies with a higher modulation of the flow structure. The ingestion mechanism is shown in Fig. 5.2. The importance of pressure asymmetries from the stationary vanes and rotating blades in main annulus which affect the rim ingestion was previously highlighted [124, 125]. High response unsteady pressure data [126] from a front cavity into a turbine stage shows the existence of coherent patterns in the rim-seal-wheel space cavity for representative turbine engine operating conditions, promoting ingress/egress flow in the under-platform cavities, as shown in Fig. 5.2 (a), which can damage the disk rim. Long-wavelength rotating structures with a size of the order of the full circumference in the inner part of the cavity, as shown in Fig. 5.2 (b), may be detected under certain conditions.

The largest structures identified in URANS simulations show these instabilities as a perturbation to the solid body rotation of the core flow created by the shear layer at the interface between the purge and the main annulus flows. While URANS models have afforded considerable insight into rotating flows such predictions involve considerable uncertainties. For instance, single-passage simulations are still used today to estimate the ingestion in turbine rim seal configurations. This approach provides misleading conclusions in terms of global ingestion since the large structures responsible for a great extent of the ingestion of hot gas are not retained in the simulations. Besides, full-annulus domains are computationally intensive and efficient methods are compulsory to reduce computational resources. Because of these difficulties, numerical

Figure 5.2.: A generic high-pressure gas turbine stage with rim seal ingestion flow (a) and rotating instabilities in the azimuthal direction at the inner part of the rotor-stator cavity core (b).

tools are not always reliable for conducting industrial designs. Computational requirements for full-annulus configurations, or a large sector of the full wheel, are even more demanding than single-passage models and this will remain an issue for the foreseeable future. Thus accurate prediction of metal temperatures will continue to rely in model calibration to compensate for effects not accounted for in the simulation. Understanding flow unsteadiness in rotating cavities has emerged as a key area of research and is the focus of this section.

The research presented here provides strong evidence of several types of unsteadiness in disk cavities using the proposed Passage-Spectral Method. In the following sections the unsteadiness of three different cavity configurations; a rotor-stator cavity, a rotor-rotor compressor cavity and a rim seal cavity is characterised. The response of different cavity modes to design variations and operating parameters is also assessed. The simulations used to validate the PSM method in cavity flows are also employed to study the various historically identified parameters that have a direct influence on the stability of rotating cavities.

5.1.1. Rotor-Stator Cavity

The flow induced by a disk rotating in a cylindrical static cavity is a simplified canonical model of configurations commonly encountered in actual turbomachinery applications. The aim of this

section is concerned with the flow instabilities in a rotor-stator cavity with a stationary casing or shroud. Confined cavity flows between two coaxial disks, where one which, or both, are rotating, are rich in unsteadiness associated with rotating instabilities that are relevant in a practical range of industrial applications. Daily & Nece [127] published flow and torque measurements for a rotating plane disk enclosed by a simple stationary shroud. This has proved a useful test case for CFD, and it was established some years ago that for the higher Reynolds numbers and cavity widths most relevant to aero-engines, conventional axisymmetric CFD models could give acceptable results when applied with care [128]. This and similar test cases have been repeated by several workers, and engine rotor-stator disk cavities are now regularly analysed both experimentally and numerically [129, 130]. In these cases the transition process is dominated by the instability of the boundary layers on the endwalls (which are thought of as disks), which results in spiral modes of very large azimuthal wavenumber. However, instabilities in terms of vortices have been found as well between the two boundary layers, at the inviscid inner part of the cavity, which rotates as a rigid body. In these two-disk flows, unsteady and non-axisymmetric instabilities at the inviscid core region will be appear [131]. In the present study, we focus on the study of these vortices in the inviscid core of the cavities, which appear under certain conditions.

Basic flows produced by rotating disks will be reviewed, and motivations for assembling this special issue dedicated to the instabilities in the inviscid core region will be presented. Despite intensive work and recent advances, no full understanding of the breakdown process has yet been achieved in these flows. Several questions regarding accuracy of all these complex cases remain. These include modelling of three-dimensional and unsteady structures, which may have an important influence on the flow into disk cavities, and may dominate over disk drag in determining the rotor moment and windage heat generation. Such features have sometimes been included in axisymmetric CFD models through the addition of body force or source terms in the momentum equations. Today, full three-dimensional CFD models are increasingly used.

The aim is to apply the proposed Passage-Spectral Method as an efficient tool to simulate this complex flows, studying the azimuthal modes which are created as a consequence of the rotating instabilities in an efficient way, simulating just a part of the full wheel domain. In the follow-

Figure 5.3.: Structured mesh and main geometrical parameters of a generic rotor-stator cavity, where $R_0 = 100\text{ mm}$ and $R_i = 20\text{ mm}$.

ing lines the geometry, the terminology adopted, and the important dimensionless parameters governing the flow together with a parametric map of the principal flow regimes are defined. However, the main focus of the section is on the instability patterns that are created and the comparison of the proposed PSM solutions against full-annulus simulations as a validation case for these complex flows.

5.1.1.1. Geometry and Flow-field Parameters

The geometrical model of the rotor-stator cavity comprises two smooth coaxial parallel disks enclosing a cylindrical domain. The sheer diversity of the resultant flows has required, however, that the present research limits attention to cases where one of the disks is at rest, where the inter-disk spacing is small compared with the disk radius, as shown in Fig. 5.3, of internal and external radius R_0 and R_i , respectively, and an axial gap of s . The upper disk of the cavity is at rest (stator) while the lower disk rotates at uniform angular velocity Ω . The surfaces of the rotor disks are highlighted with red lines, which spin about the rotational axis, while stator disks are highlighted with blue lines. Every rotating and non-rotating surface is imposed to be adiabatic.

This geometry and flow field are fully characterised by two parameters, the aspect ratio of the cavity, defined by $G = s/R_0$, and a global parameter which may be taken as the rotational

Figure 5.4.: Reduced-passage sample mesh of the rotor-stator cavity with a sector size of 6° .

Reynolds number based on the external radius, defined by

$$Re_\phi \equiv \Omega R_0^2 / \nu \quad (5.1)$$

The Reynolds number based on the gap can be defined as $Re_s \equiv \Omega s^2 / \nu$, where $Re_\phi = Re_s / G^2$ [132]. The present research limits attention to the case of radially elongated cavities ($G < 1$) because of their relevance in gas-turbine systems. For small gap widths, a strongly viscous affected shear flow develops while for higher axial distances, a flow with two boundary layers and a core region establishes.

Geometrical parameters of the rotor-stator cavity are listed in Fig. 5.3, where the gap between the stationary and rotating parts, s , is modified in order to study the effect on the flow of the aspect ratio of the cavity G . The computational meshes are generated by first defining a 2-D mesh in the plane (x, r) and then extruding this circumferentially. A view of the structured grid [107] is outlined in Fig. 5.3 and Fig. 5.4. The grid of the reduced-passage sample, which will be used to carry out PSM solutions, is shown in Fig. 5.4, where the sector size is 6° . The mesh of each reduced-passage consists of about 8.96×10^4 nodes. The full-annulus domain consist then in a number of $N_B = 60$ reduced-passages to reconstruct the whole annulus, and then the entire mesh is approximately 5.38×10^6 nodes.

Figure 5.5.: Sketch of the flow pattern in the inner part of a rotor-stator cavity.

5.1.1.2. Steady Flow-field

First, the steady and axisymmetric flow-field in between two coaxial disk, where one is stationary and the other is rotating is briefly described in this section are extensively studied in Appendix B. A similarity solution was found by [133] solve the steady, rotationally periodic equations of motion qualitatively for the flow between a rotating and a stationary disk with infinite radial extent where the flow is at rest. The similarity principle also applies if the fluid far away from the wall rotates as a rigid body [134] and even in the case of two infinite coaxial rotating disks, which is actually our case. In the inner region of the cavity, the flow-field agree very well with the self-similar solution [135, 136]. Figure 5.5 shows schematically the flow pattern consisting of two separated boundary-layers attached to both disks for $Re_\phi \gg 1$, where $\delta_c \sim (\nu/\Omega)^{1/2}$ is the characteristic thickness of the disks boundary-layers, being ν the kinematic viscosity. The core region is characterised as a rigid body rotation, where the fluid is rotating at a swirl speed of $\beta \simeq 0.3$ [137], where $\beta = \omega_{core}/\Omega$ and ω_{core} is the angular velocity of the fluid in the core of the disk. The circumferential velocity in the inner region varies linearly with the radius, which corresponds to the value expected for a high Reynolds laminar flow and large clearance (Regime II according to Daily et al. [127]). The radial velocity is nearly zero in the core region, $v_r \simeq 0$, and the axial velocity is small in comparison with the azimuthal velocity, $v_\theta \ll v_z$.

The unsteady behaviour of this solution appears in the inviscid core region, when the flow becomes non-axisymmetric in the azimuthal direction. It can be demonstrated, as analysed also in Appendix B, that the non-dimensionalised characteristic time in rotor-stator cavity, $\tau = \Omega t$, scales as $\tau_c \sim Re_s^{1/2}$. Having this into account, typically the acceleration time of a large

gas turbine is then of the order of a few seconds. If we assume also that the flow in finite rotor-stator cavities very rapidly develops non-axisymmetric instabilities in the inviscid core leading to laminar behaviour, it could be concluded that the use of a quasi-steady approach is questionable for this kind of application an efficient numerical methods to simulate unsteady three-dimensional domain for this kind of rotating cavities is required.

In order to define the geometric and flow-field parameter to study the unsteady behaviour of the solution into an enclosed rotor-stator cavity previously defined, Schouveiler et al. [3] mapped the transition to unsteady and non-axisymmetric behaviour of the flow in terms of gap ratio and Reynolds number is used as a reference, as shown in Fig. 5.6. There is a large disparity of results in the literature linked to various properties of these instabilities. Besides, a large variety of structures can be observed according to the combination of G and Re_ϕ , but can be summarized in Fig. 5.6, where coloured lines represent the boundary between stable solutions (steady and axisymmetric behaviour in the azimuthal direction) and unstable solutions (unsteady and non-axisymmetric behaviour in the azimuthal direction). In terms of the overall flow, these instabilities cause waviness in the streamlines, but as the disturbances have vorticity, they appear as travelling vortices rolling up around a circular or spiral axis when the basic flow is subtracted from the total [131].

5.1.1.3. Results

First, a comparison between a stable and unstable solution is outlined. Steady solution of a two-dimensional mesh domain (x, r) , corresponding to points [a] and [b] in Fig. 5.6 are used as the initial solution of the unsteady full-annulus simulation (360° model). The Reynolds of this problem based in Eqn. 5.1 is $Re_\phi = 4.3 \times 10^4$. These simulations are conducted with 1000 time steps per rotor revolution. The number of pseudo time steps per physical time step was set to 300, which is large enough to achieve the convergence criterion of the inner iteration of the dual time step for this type of flows.

Figures 5.7 (a) and (b) show the comparison of full-annulus solutions for cases [a] and [b] in Fig. 5.6 respectively, in terms of dimensionless radial velocity on the mid-cavity axial plane

Figure 5.6.: Transition diagram by Schouveiler et al. [3] for large clearance and laminar flow. Curves labeled 'CR' indicate threshold for circular rolls, where the solution becomes non-axisymmetric.

Figure 5.7.: Dimensionless radial velocity contours on the mid-cavity axial plane for a stable and unstable case, corresponding to points [a] and [b] in Fig. 5.6 respectively, after 20 rotor revolutions.

after 20 rotor revolutions. As can be observed in Fig. 5.7 (a), the stable solution case [a] in Fig. 5.6, shows a uniform contour in the azimuthal direction. The unsteady solution of three-dimensional full-annulus domain for this stable case remains axisymmetric along time. On the contrary, the unstable case defined as case [b] in Fig. 5.6, presents a non-axisymmetric pattern in the azimuthal direction, as shown in Fig. 5.7 (b) for the radial velocity solution at the core region. These two full-annulus solutions confirm the stability analysis mapped by Schouveiler et al. [3] in Fig. 5.6, representing a three-lobed structure of low radial velocity patterns in the azimuthal direction. The question now arises if the proposed Passage-Spectral Method with a certain number of spatial harmonics can reproduce this non-axisymmetric unsteadiness.

A comparison between the full-annulus (360° model) and PSM computations will be given next. Figure 5.8 (a) shows a full-annulus computation domain, previously outlined in Fig. 5.7 (b), while Fig. 5.8 (b) represents the solution obtained by the proposed PSM with four spatial harmonics ($M = 4$). The contour plots of the dimensionless radial velocity show same three-lobed pattern, referred as a third spatial harmonic ($m = 3$). Although there are small differences in terms of the amplitude of the instability, the method is able to accurately reproduce the associated instability due to the inherent unstable behaviour of the case. The structures visualized in Figs. 5.8 (a) and (b) are assuredly permanent, and exhibit a quasi-steady converged state for a long computation time. These structures present an associated unsteadiness and are characterised by a fraction of the rotational speed of the rotating wall $\Omega_c < \Omega$, as outlined in the following lines.

5.1.1.4. Frequency Spectrum

In order to further study the final state of the unstable solution obtained in Fig. 5.8, the synchronous response of the radial velocity at the mid-cavity axial plane is used to apply the fast Fourier transform (FFT) to the unsteady data, and the results are plotted in Fig. 5.9, where the frequency spectrum of the third spatial harmonic of the signal is outlined. Seen for the rotor frame of reference, the structures are in a traveling circumferential traveling wave form, moving relative to the rotor disk at a fraction of the rotor speed. Figure 5.9 shows then inherent three

Figure 5.8.: Comparison of dimensionless radial velocity contours on the mid-cavity axial plane for full-annulus and PSM ($M = 4$) solutions after 20 rotor revolutions. $G = 0.05$, $Re_\phi = 4.3 \times 10^4$.

unsteady structures in the core region rotating at about $\sim 50\%$ of disk speed, where the horizontal axis represents the angular velocity of propagation of the harmonic relative to the rotor, $\Omega_m = \omega/m$, non-dimensionalised with the rotor speed, Ω_{rotor} , where m is the spatial harmonic index of the rotating instability, which in this case is a three-lobed structure and then $m = 3$. This velocity of the rotating instability in the core region of the cavity can be compared with the one obtained by Czarny et al. [138], who carried out simple flow-visualization experiments in a rotating water table for empty rotor-stator disk cavity where quite unexpected three-lobed large-scale vortices rotates at roughly half the speed of the rotor.

For a longer computation time, the transient development of both solutions is expected to be identical. In conclusion, the comparison of the full-annulus and PSM case proves that the use of the proposed method with four spatial harmonics appears to be sufficient. The excellent agreement between the two solutions clearly demonstrates the validity of the present spatial Fourier-transform-based approach.

We can then conclude that the PSM with four spatial harmonics ($M = 4$) can reproduce the unsteadiness associated with a rotor-stator cavity rotating instability with a reduction in computational time and resources. The speed-up factor of the PSM, using Eqn. 2.13 with $M = 4$ and $N_B = 60$, is $R = 60/9 \simeq 7$.

Figure 5.9.: Comparison of non-dimensional relative angular phase velocity of the third spatial harmonic, $m = 3$, of the radial velocity between PSM ($M = 4$) and full-annulus solutions at a numerical probe located on the mid-cavity axial plane for the rotor-stator cavity. $Re_\phi = 4.3 \cdot 10^4$; $G = 0.05$.

5.1.2. Compressor Cavity

Axisymmetric flows in co-rotating disk cavities with a radial inflow or outflow of fluid were investigated in considerable detail in the 1970s and 1980s [132]. Such flows are regularly calculated using CFD in industry and models are increasingly extended to include three-dimensional features such as holes or protuberances. However, particular difficulties have been encountered in co-rotating disk cavities with little or no imposed radial throughflow. Such cavities occur in high pressure compressors in which the disk rims are generally hotter than the disk cobs, which are cooled by a central axial throughflow. Sealed annular cavities have also been used in some engines. The positive radial temperature gradient results in strong buoyancy effects in the centripetal force field. Experimental configuration of a sealed rotating annulus heated at the outer shroud and cooled at the inner shroud has been studied with CFD by several authors [139, 140, 141], who performed unsteady laminar flow computations and found irregular fluctuations in the flow.

Any radial disturbance would be subject to the influences of centrifugal effect and density variation due to the temperature for buoyancy-induced flows in rotating cavities [28, 4]. In order for a sustained radial flow pattern to exist there needs to be a mechanism to sustain both radial inflow as well as radial outflow. In terms of momentum balance, a radial inflow in the rotating

Figure 5.10.: High-pressure aeroengine compressor and schematic diagram of open rotating cavity [4].

system will need a Coriolis force in one circumferential direction whilst a radial outflow will need the Coriolis force in opposite direction. A pattern to fit these requirements is a pair of contra rotating vortices in the core region of the cavity. The directions of the circumferential pressure differences are exactly those required by the Coriolis effects to maintain the opposite radial movements of the vortices. The strength of the radial movement is directly linked to the strength of the vortices, which in turn influence the magnitude of the circumferential pressure difference and thus the Coriolis forces. The resulting flows have been known to be strongly three-dimensional and unsteady for some time and have been recognized as particularly challenging for CFD, since full-annulus geometries are needed in order to capture these rotating structures in the azimuthal direction.

The proposed Passage-Spectral Method is used then to facilitate efficient computational predictions of the non-axisymmetric flows in such compressor cavity configurations. Given that the flow non-uniformities in the circumferential direction are of large length scales, only a few circumferential Fourier harmonics would be needed, resulting in a drastic reduction in number of circumferential mesh points to be required.

Figure 5.11.: Computational mesh and main geometrical parameters of a generic compressor cavity, where $R_0 = 400\text{ mm}$ and $R_i = 140\text{ mm}$ and the gap between coaxial rotating disks is $s = 80\text{ mm}$.

5.1.2.1. Geometry and Flow-field Parameters

The geometry considered in this section is based on the rig described by Bohn et al. [140] and He [62], where the experimental and computational flow visualization reveals that different patterns of vortices appeared periodically. The cavity geometry values and mesh are illustrated and listed in Fig. 5.11. Every wall rotates at a uniform angular velocity Ω , as shown in Fig. 5.11.

The cavity shroud wall temperature is specified at 355K. The hub surface temperature is specified at 300K, which is also the stagnation temperature of the axial inlet flow. The two disk surfaces are specified adiabatic. The axial inlet and exit annulus duct wall surfaces are also subject to a specified total temperature of 300K. The Mach number at the inlet is imposed to be $M_0 = 0.02$.

The computational meshes were generated by first defining a 2-D mesh in the plane (x, r) and then extruding this circumferentially. He [62] used a 2-D mesh to simulate this case due to the axisymmetric nature of the mesh. Here a sector of 10° is used because the generalization of the Fourier method to non-axisymmetric geometries. The results can be then obtained also with a 2-D mesh as done in the work presented by He [62]. Figure 5.12 shows the reduced-passage

Figure 5.12.: Reduced-passage sample mesh of the compressor cavity with a sector pitch of 10° .

computational mesh used for the PSM simulation, instead of full-annulus geometry, where the pitch of the sector domain is 10° . Each reduced-passage consists of about 3.28×10^5 nodes per passage. The size of the full-annulus mesh is then approximately 11.8×10^6 nodes, consisting on a number of $N_B = 36$ reduced-passages to reconstruct the full wheel domain.

5.1.2.2. Results

First, a comparison between a stable and unstable solution is outlined. Steady solution of a two-dimensional mesh domain (x, r) are used as the initial solution of the unsteady full-annulus simulation (360° model). These simulations are conducted with 800 time steps per rotor revolution. The number of pseudo time steps per physical time step was set to 200, which is large enough to achieve the convergence criterion of the inner iteration of the dual time step for this type of flows.

Figures 5.13 (a) and (b) show the comparison of full-annulus solutions for two different operating conditions, in terms of dimensionless radial velocity on the mid-cavity axial plane. As can be observed in Fig. 5.13 (a) for the stable solution case, the radial velocity contour in the azimuthal direction presents a uniform behaviour for a rotational Reynolds number $Re_\phi = 4.2 \times 10^5$ (see Eqn. 5.1). The unsteady solution of three-dimensional full-annulus domain for this stable case remains axisymmetric along time. On the contrary, the unstable case in Fig.

Figure 5.13.: Dimensionless unsteady radial velocity contours at mid-cavity compressor axial plane after 30 rotor revolutions.

5.13 (b) presents a non-axisymmetric pattern in the azimuthal direction for a rotational Reynolds $Re_\phi = 1.95 \times 10^6$ according to Eqn. 5.1. After 30 rotor revolutions, the 360° domain unstable solution produces a sharp increase in the spatial harmonics modes in a numerical probe located in the mid-cavity axial plane, which indicates a rapid development of the non-axisymmetric flow with significant radial motion. This is in agreement with the conclusions outlined by Owen [28], where this non-axisymmetric unsteadiness can affect the enhanced convection heat transfer due to the large scale radial fluid transportation defined as Rayleigh-Bénard flow. There is, however, a noticeable discrepancy between the results obtained for the unstable case in Fig. 5.13 (b) and the previous computational results outlined by He [62]. The sensitivity of the flow pattern (number of vortex pairs) to the form of the inlet disturbances of small amplitudes is further revealed by He [62]. It is observed for large number of rotor revolutions that the first unstable spatial harmonic is the third one, during the first 10 rotor revolutions after the unsteadiness is triggered. However, as previously concluded by other authors [140], a cyclically transitional pattern with one-two-three-two-one pairs of contra-rotating vortices was observed in the simulations.

A comparison between the full-annulus (360° model) and PSM computations will be given next. Figure 5.14 (a) sketches a full-annulus computation domain, previously shown in Fig. 5.13 (b), while Fig. 5.14 (b) represents the solution obtained by the proposed PSM with six spatial harmonics ($M = 6$). The PSM solution indicated that the PSM solution is no longer harmonic-

Figure 5.14.: Comparison of radial velocity contours on the mid-cavity compressor axial plane for full-annulus and PSM solutions after 30 rotor revolutions. $Re_\phi = 1.95 \times 10^6$.

dependent and six spatial harmonics suffice to predict full-annulus solutions of three vortex pairs ($m = 3$). The occurrence of a non-axisymmetric flow is captured by the present Fourier solution with a few number of reduced-passages samples. Although the vortices are oscillatory in time and a difference in phase of the instantaneous location of the vortices can be appreciated comparing Figs. 5.14 (a) and (b), no pattern change in terms of number of instability modes has been observed along time. The structures visualized in Fig. 5.13 (b) and Figs. 5.14 (a) and (b) exhibit a quasi-steady converged state for a long computation time.

The above comparisons between the Fourier model and the full-annulus solution are encouraging, illustrating the potential capability of predicting efficiently the non-axisymmetric flow characteristics in this rotating cavity configuration. In order to increase the credibility of the results, the quasi-steady state of both solutions, PSM and full-annulus simulations, are compared in the following section studying the associated unsteadiness of these structures.

5.1.2.3. Frequency Spectrum

Figure 5.15 shows the frequency spectrum of the third spatial harmonic ($m = 3$) associated with the rotating velocity speed of the three vortex pair in the core region of the compressor cavity, represented by monitoring the radial velocity, which counter-rotates at about $\sim 95\%$

Figure 5.15.: Comparison of non-dimensional relative angular phase velocity of the third spatial harmonic, $m = 3$, of the radial velocity between PSM ($M = 6$) and full-annulus solutions at a numerical probe located on the mid-cavity axial plane for the compressor cavity. $Re_\phi = 1.952 \cdot 10^6$.

of disk speed for the first 10 rotor revolutions after the unsteadiness is triggered, where the horizontal axis represents the angular velocity of propagation of the harmonic relative to the rotor, $\Omega_m = \omega/m$, non-dimensionalised with the rotor speed, Ω_{rotor} . It was observed by other researchers [28] that the vortex core would typically slip relatively to the rotating disk surface at a small fraction (e.g. about 10%) of rotation speed, which is approximately the results obtained here. PSM is able to reproduce not only to reproduce the trigger of the instability, also accurately capture the unsteady behaviour of the rotating instabilities in a compressor cavity configuration.

The important conclusion here is that the method accurately reproduces the full-annulus frequencies associated with the three vortex pair of the solution. We can conclude that the PSM with four spatial harmonics ($M = 6$) can reproduce the unsteadiness associated with a compressor cavity rotating instability. The speed-up factor of the PSM, using Eqn. 2.13 with $M = 6$, is $R = 36/13 \simeq 3$.

5.1.3. Rim Seal Cavity

The internal air system of industrial gas turbines and aero engines has to avoid the ingestion of hot gas from the main annulus into the stator/rotor wheel-space. As a consequence cooling air, diverted from the compressor, must be used to keep the hot gas off the vulnerable rotating com-

ponents. The cooling air returns to the main turbine gas path through a rim seal clearance. Use of excessive cooling air will result in additional losses, which will reduce the efficiency of the gas turbine. Since the reduction of the cooling air consumption leads to a direct benefit for the engine performance, it is essential that the required cooling air massflow is accurately predicted. Rotationally-driven ingestion is associated with the disc pumping effect, while pressure-driven ingestion is related to the pressure asymmetry caused by vanes and blades in the main gas path. These pressure asymmetries in the main flow result in inflow and outflow through the rim seal, as schematic shown in Fig. 5.2. This unsteadiness is deterministic by higher frequency fluctuations caused by perturbations with a length-scale of the order of a blade pitch. In addition to these mechanisms, large-scale unsteady flow modes with characteristic frequencies unrelated to those of the rotating blade in the main annulus have been reported recently, as in the review paper by Chew et al. [142].

In the presence of the intrinsic unsteady flow modes empirical correlations and Reynolds-averaged Navier-Stokes (RANS) models give solutions for sealing effectiveness that depart from the experimental measurements [120, 143]. These cavity modes were dominant compared with the blade passing mode, and that the strengths and characteristic frequencies of the cavity flow structures were suppressed as the purge mass flow was increased. In some cases the intrinsic unsteadiness is confined to a quite small rim seal region, in other cases unsteadiness (particularly lower frequency components) propagates further into the disk cavity and/or annulus, possibly interacting with blade or vane passing. However, even with fully axisymmetric and steady boundary conditions, the flow was predicted to be three-dimensional and unsteady. Gao et al. [25] included the disk cavity and the main flow annulus on their CFD model but did not include rotating blades or stationary vanes, and rotating flow structures appear in the azimuthal direction near the rim seal region. Apart from an externally imposed three-dimensionality, the flow in a cavity can become three-dimensional due to the formation of vortex structures. This effect has been observed to be strongest and clearest at high purge flow rates [144]. The shear layer instability could clearly affect mixing of the two streams in the annulus although its role in low purge flow, ingesting conditions is less clear. Some researchers also studied the influence

of the rotor blades and vanes into these rotating instabilities, shown that in the absence of blades these structures were still shown to occur [123]. Despite intensive work and recent advances, no full understanding of the breakdown process has yet been achieved in these flows. The aerodynamic interactions and heat transfer between the through-flow and the cavity flow will affect the pressure losses, flow rate as well as the temperature of the cooling flow. The problems that are associated with front cavities and low cooling air mass flow were subject to an extensive numerical study. Chew et al. [142] gave thorough review of the flow mechanisms in axial turbine rim sealing and stability of rotor/stator cavities, summarizing many of the experimental and computational studies that have identified non-blade passing related unsteadiness believed to be associated with the rotating flow modes.

Nevertheless the investigated URANS simulation approach using single-passage domains or sector models still show significant deviation in the prediction of the hot gas ingestion compared to the experimental test data [22, 121, 145]. These numerical approaches are not capable of resolving sufficiently characteristic flow phenomena inside the wheels space and the sealing gap. This is due to the fact that flow phenomena in circumferential direction, resulting in rotating fluctuations generated in rim seal cavities, are purely non-axisymmetric in the full wheel space and full-annulus geometries are they needed to accurately resolve length-scale of the order of the rotor circumference. These type of flow can arise into a time-dependent cavity flow which is the case chosen in this research to show the potential of the proposed method into a non-axisymmetric geometry such as rotor-stator cavity including the row of rotor blades. The proposed Passage-Spectral Method with a few number of spatial harmonics aims to capture these rotating instabilities. To do that a comparison against full-annulus solution will validate the novel approach.

5.1.3.1. Geometry and Flow-field Parameters

The method is verified on a high-pressure turbine stage consisting on 78 stator/rotor cavities with the row of rotor blades. Schematics and some geometrical parameters are shown and listed in Fig. 5.16. The surfaces of the rotor disks are highlighted with red lines, which spin about the

Figure 5.16.: Computational mesh and main geometrical parameters of a generic turbine rim seal cavity, , where $R_0 = 150\text{ mm}$ and $R_i = 40\text{ mm}$, $s = 9\text{ mm}$ and the axial gap pf the rim seal is $b = 3\text{ mm}$.

rotational axis, while stator disks are highlighted with blue lines. The reduced-passage sample to simulate the proposed Passage-Spectral Method uses one rotor airfoil into the sector domain. In this turbine, the disk rim gap under consideration consists of a simple axial gap between the rotating disk. The angular velocity of the rotor is 10624 rpm. A total mean pressure boundary condition ($P_0 = 2.25\text{ bar}$) at the inlet is used at the gas path simulating the effect of the previous row with a total temperature of $T_0 = 414\text{ K}$. A static pressure of $P_s = 1.73\text{ bar}$ is imposed at the outlet of the rotor blade. The corresponding mass flow for a 360° degree geometry would be $\dot{m}_{inlet} = 4.96\text{ kg/s}$. The cooling air supplied to the hub region is defined by the non-dimensional cooling air mass flow rate, C_w , defined by

$$C_w = \frac{\dot{m}_{cooling}}{\mu \cdot R_0} \quad (5.2)$$

μ is the dynamic viscosity. The mass flow imposed at the cooling flow, $\dot{m}_{cooling}$ as shown also in Fig. 5.16 in the hub region of the cavity, is imposed with a temperature of $T_{cooling} = 300\text{ K}$. For this case $C_w = 2 \times 10^4$. The standard $k - w$ turbulence model with wall functions is used and every wall is imposed to be adiabatic. The Reynolds of this problem based in Eqn. 5.1 is $Re_\phi = 1.25 \times 10^6$. A converged steady-state single passage model solution was used initially. A

Figure 5.17.: Reduced-passage 1/78 th sector mesh of the rim seal cavity configuration.

complete engine revolution was first computed as initial condition, then calculations are run for an unsteady case with 800 time steps per rotor revolution. As we will show in the following lines, the rim seal flow is dominated by unsteady effects under this operating conditions.

The mesh grid of the reduced-passage sample used in the PSM resolution is shown in Fig. 5.17, which consists of about 5.72×10^5 nodes. In order to capture the complex flow behaviour right at the sealing lip, a high resolution in this region was implemented. Adding the row of rotor blades the mesh turns into non-axisymmetric in the azimuthal direction. This geometry then can only be simulated using spatial Fourier series if the method is extended to a block-wise numerical resolution, as done in this thesis. The three-dimensional, unsteady CFD analysis using the proposed PSM is performed then with a 78-th sector model of the annulus. The size of the full-annulus mesh is then approximately 44.7×10^6 nodes. Figure 5.18 schematic displays five reduced-passages required to carry out a spatial Fourier transformation for two spatial harmonic using the proposed method. To simulate PSM with M harmonics, we need then to use $2M + 1$ reduced-passages, represented in Fig. 5.17, uniformly spaced along the whole wheel domain, where θ_j in Fig. 5.18 refers to the azimuthal location of each j -th block or reduced-passage.

Figure 5.18.: Schematic distribution of five j -th reduced-passage samples used by PSM with $M = 2$.

5.1.3.2. Results

The overall objective of this study into the rim seal configuration is to determine if the proposed Passage-Spectral Method is able to predict the unsteadiness associated with rotating instabilities in the inner part of the cavity. The numerical resolution of large scale rotating structures that is found in the stator/rotor cavity is a special focus of this section. Snapshots of instantaneous radial velocity contours are shown in Fig. 5.19, extracted on an axial plane mid-way between stator and rotor walls. The resulting flow structures are fully three-dimensional and time-dependent. Figures 5.19 (a) and (b), for full-annulus and PSM ($M = 14$) solutions respectively, show multiple recirculation cells to occur in the seal region flow after few rotor revolutions from the initially converged steady-state single passage solution. Self-induced development of the non-axisymmetric of the flow was achieved with any kind of external disturbance due to the inherited instability of the fluid under this operating conditions. Signatures of large-scale unsteady flow structures are observed at the axial seal clearances and the inner part of the cavity for both solutions in Figs. 5.19 (a) and (b). The reason of the discrepancy between PSM solution with $M = 14$ in Fig. 5.19(b) and full-annulus solution in Fig. 5.19 (a) is the inaccurate resolution of higher spatial harmonics associated with the unsteadiness of the instabilities in the inner part of the cavity. However, as we will show in the following lines, an accurate prediction of the ingestion in the rim seal region can be captured with this number of spatial harmonics in the

Figure 5.19.: Comparison of unsteady radial velocity contours at mid-axial plane of the turbine stage stator/rotor cavity for full-annulus and PSM solutions after 20 rotor revolutions.

proposed method. Similar large scale patterns inside turbine rim cavity was previously reported by Cao et al. [22] and Jacoky et al. [144]. However, this phenomenon cannot be captured with steady simulation or a sector mesh. The use of periodic boundary conditions suppresses all frequencies in the flow, where the wavelength is not a multiple of angular sector size times radius. In addition, as shown in Fig. 5.19 (c), for the simulation of PSM with $M = 6$ the solution remains axisymmetric near the rim seal region along time, although the same radial arms can be shown at the three solutions in Fig. 5.19 and be captured with six spatial harmonics. With this number of spatial harmonics resolution the unsteadiness at the rim seal is suppressed. The method is filtering higher harmonics than $M = 6$, and this is the ultimate reason why in the inner part of the cavity the instability is not triggered. The number of harmonics of these fluctuations is decoupled from the rotor 78th harmonic.

Although this low harmonic model could indicate the onset and occurrence of non-axisymmetric flow, enough number of spatial harmonics need to be solve with the proposed method since a low number of harmonics could influence the solution. The calculated instantaneous circumferential pressure distribution in the rotor wall at the inner part of the cavity for PSM ($M = 14$) solution is shown in Fig. 5.20. It is important to highlight that this phenomenon is captured with the proposed method just simulating a certain number of uniformly spaced $2M + 1$ reduced-passage samples along the circumferential coordinate, as sketched in Fig. 5.18 for $M = 2$. These pressure asymmetries captured by the method result in inflow and outflow through the

Figure 5.20.: Instantaneous static pressure field on the rotor wall of PSM ($M = 14$) solution after 20 rotor revolutions.

rim seal. Where the pressure in the cavity is locally lower than the main annulus pressure, ingestion occurs. Regions of low pressure are leading to local ingestion of hot gas into the stator/rotor wheel-space (negative radial velocities in Fig. 5.19). The pressure gradient is then a measure for hot gas ingestion. The existence of this flow pattern persists within the imposed range of cooling air and significantly affects the pressure distribution as shown in Fig. 5.20. The occurrence of a large scale rotating structures within the cavity can be accurately captured by a time-domain Fourier method applied to a reduced-passage model. The use of periodic boundary conditions or time periodicity methods in these complex configurations cannot accurately predict the non-axisymmetry in the flow pattern within the cavity and the interaction with the blade row.

Figure 5.21 shows the time development of the dimensionless temperature with respect the hot temperature at the mainflow inlet at a numerical probe in the inner part of the cavity and near to the rim seal in the rotor frame of reference, represented in Fig. 5.20. The unsteadiness effect modifies the predicted temperature fluctuations in the numerical probe. Thus, the frequency of these fluctuations is decoupled from the rotor frequency, and rotated with its own associated frequency in the rotor frame of reference, as shown in the history of the temperature at the

Figure 5.21.: Comparison of ingestion flow rates between full-annulus and PSM ($M = 14$ and 6) solutions monitored at the numerical probe in Fig. 5.20.

numerical probe in Fig. 5.21. During the first three rotor revolutions, full-annulus and PSM solutions, with $M = 14$ and 6, remains stable and no fluctuations has occurred yet. After about three revolutions, the temperature rate begins to fluctuate around 30% for full-annulus and PSM ($M = 14$) solutions as a consequence of the ingestion of hot temperature at the mainflow. A quite stable pattern can be observed after 8 rotor revolutions, and this mean value stays at the same level until the calculation was terminated at 20 rotor revolutions. At the rim gap, alternate regions of cavity inflow and outflow occur around the circumference. The mean value temperature ingestion at the numerical probe about 15% of the hot temperature at the inlet is comparable in PSM with $M = 14$ spatial harmonics and full-annulus solutions. The main conclusion is that the unsteady analysis of the PSM solution with spatial harmonics higher than $M = 14$ does not produce a significant change in the answer of temperature ingestion in the rim seal.

This implies that although low harmonics solutions do not reproduce exactly the full-annulus solutions in quantitatively describing these complex flows richer in spatial harmonics, PSM solutions might be used to indicate the onset and occurrence of an non-axisymmetric flow with a significant reduction in computational time and mesh size. The number of spatial harmonics used in the PSM solution have been chosen to provide the best compromise between a sufficient level of accuracy in comparison with full-annulus solution and the least computational effort.

With respect to the saving in computational resources, Tab. 5.1 compares the computational

	<i>PSM</i> ($M = 14$)	<i>FULL</i>
Number of points	16×10^6	44×10^6
Number of GPUs (GTX 1080 Ti)	3	9
Wall clock time per revolution (h)	1.1	1

Table 5.1.: Computational cost of different numerical approximations for rim seal configuration cavity for a mesh of 0.57×10^6 nodes per reduced-passage sample.

effort requires for the two solutions. The speed-up R in computational mesh points in the circumferential direction for 3D non-axisymmetric domain geometry in comparison with a full-annulus solution with 14 harmonics in the spatial Fourier transform will be then $R = 78/29 \simeq 3$, with $N_B = 78$. The Passage-Spectral Method cost is significantly smaller and requires only $1/R \simeq 1/3$ of the number of GPUs. The exact benefit of the method is strongly case dependent and varies with the desired order of approximation. The more number of blades the higher the potential reduction in computational cost.

5.2. Conclusions

This chapter presents the application of the PSM to predicting rotating instabilities in the inviscid core region of three different rotating cavity configurations. The accuracy of the method is verified against full-annulus simulations. The need of fine meshes large sector models makes this type of simulations expensive to perform, and create the need to develop efficient numerical methods. The artificial imposition of periodicity in small-sized sectors may impose restrictions for the development of large-scale flow structures. The Passage-Spectral Method is then proposed for simulating this complex flows, retaining a few spatial harmonics in order to capture the large scale structures in the azimuthal direction. The Fourier solutions with a certain number of sectors indicate how many spatial harmonics are required to properly resolve each pair of contra-rotating vortices and rotating instabilities.

Simplified rotating cavities are used to validate the proposed method as well as to shed some light about the non-axisymmetric and unsteady behaviour of large-scale structures which occur in the rotating core of the cavities under some operating conditions. The aim is to characterise

the rotating circumferential modes in the inviscid core region of different cavities. As the flow non-uniformities in the circumferential direction have long wave-lengths, only a few circumferential harmonics are needed, resulting in a drastic reduction in number of mesh points required for such analyses. Computational results of the present Fourier model agree well with those from the direct whole domain solutions. Large scale rotating structures that revolves with 50% of the rotor speed are detected in the rotor-stator cavity using PSM with four spatial harmonics, in good agreement with the full-annulus results. For the rotor-rotor compressor cavity, the frequency spectrum shows that the PSM solution with six spatial harmonics and the full-annulus simulation, capture the rotational speed of a three-lobbed structure at 95% of the rotor speed. In the rim seal cavity configuration, the rotating forcing in the cavity rim significantly influences the ingestion of hot gas into the turbine wheel space, as shown in the non-axisymmetric and amplitude behaviour of the radial velocity. Single passage methods or sector models can not capture the unsteadiness associated with rotating instabilities in rotating cavities, underestimating the ingestion of hot gas into the turbine wheel space. Full-annulus simulations, or as proposed in this research, the use of the PSM with a high enough number of spatial harmonics, are needed to accurately solve the flow-field.

It is safe to say that the demands in processing power for URANS calculations for wheel space performance evaluations is still high and prevent the systematic use in a design environment. The spatial Fourier solutions with different number of harmonics indicate that a few number of spatial harmonics are required to resolve quantitatively each pair of contra-rotating vortices with accuracy and enable the trigger of the instabilities of interest. Advances in the understanding of the related unsteady and non-axisymmetric phenomena in different rotating cavities have been achieved based on the use of the proposed method on simplified cavity models.

Further, application of the same methodology with a low number of spatial harmonics to several realistic gas turbine rotating cavities may confirm the perceived knowledge and received agreements from experimental indications. In the rim seal cavity configuration, more spatial harmonics are needed to reproduce the unsteadiness in the rim seal region. Nevertheless, the Fourier solution with a few harmonics has shown for the first time that the method can cap-

ture the non-axisymmetric behaviour associated with low spatial modes in the inner part of the cavity.

6. Conclusions

A novel block-wise spatial Fourier approach has been introduced and verified within this thesis. After a detailed introduction into the theory of the proposed method and its implementation, the method has been successfully applied to realistic turbomachinery flows which are characterised by their spatial cyclic symmetry, without making any hypothesis about its temporal periodicity. The method is able to filter the unwanted harmonics and facilitate efficient computational predictions of non-axisymmetric and unsteady flows for a quite wide range spatial harmonics spectrum of interest.

The proposed method is intended to problems with a short characteristic length perturbation with a much larger superimposed wavelength wave. The long wavelength perturbation is accurately captured by a discrete Fourier transformation in the periodic boundaries of each block, while the short wavelength perturbation is embedded inside each reduced-passage or block and it is not affected by any Fourier transform. However, it is demonstrated that the proposed block-wise spectral reconstruction is able to capture the interaction between low spatial harmonics with higher harmonics of the short wavelength correctly. The results of the proposed approach carried out with one-dimensional discrete Fourier transforms in the resolution of the linearised wave equation leads to good agreement against full-length and analytical solutions and confirms the overall existence and influence of scattered spatial harmonics accurately captured by the method. The proposed PSM is found better than initially expected, since Tyler-Sofrin modes are natively retained in the block-wise reconstruction, which makes the method very well suited for turbomachinery problems. The method is intended to be used to compute flows exhibiting large-scale instabilities, which gives rise to a higher speed-up factor since we only need a few

blocks to approximate the exact full-length solution. This thesis extends this novel strategy for unsteady flows, applied for the first time to non-axisymmetric geometries using spatial Fourier series, where the fundamental frequencies of the problem are not known beforehand.

The proposed methodology can produce reliable predictions of the non-linear stability of a generic fan-stage under forced response due to inlet distortion and rotating stall instabilities with a few number of spatial harmonics retained in the method, by comparing the results against full-annulus solutions. In addition, the effect on the non-linear stability of a fan-stage imposing complex inlet and outlet non-axisymmetric pressure patterns is assessed by slightly modifying the method, aiming at handling inlet distortion screens in a nodal diameter by nodal diameter basis, just retaining the spatial harmonic of interest. It is concluded that performing a non-linear simulation with the proper single harmonic perturbation is enough to assess fan stability without recurring to full-annulus simulations. The non-linear contribution of each harmonic can be assessed in a separate way. Stability analysis of distorted inflows showed that the first nodal diameter is the most unstable and exhibits a sub-critical bifurcation, whilst the third nodal diameter has a stable contribution and exhibits a super-critical bifurcation. This is also the case for the stability analysis for potential perturbations downstream of the fan, where all the analyzed nodal diameter present a super-critical bifurcation and a stable contribution to the fan-stage stability. Besides, studying the final saturated state of unstable cases, it can be concluded that by increasing the amplitude of the inlet disturbance the non-linear contribution of the higher harmonics to the final saturated state is increased as well. This study not only shows a reduction in computational time to assess non-linear fan stability by a factor of seven but also creates an efficient methodology for understanding the non-linear instability of fans due to inlet and outlet distortion profiles, accurately predicting rotating instabilities in fan-stages.

Different configurations of rotating cavities with their associated frequencies have been solved as an application case using the proposed method with a low number of spatial harmonics. Rotating instabilities in aero-engines are difficult to accurately resolve for numerical simulation because of its complex flow physics. However, it is been shown that PSM produces confident approximation to full-annulus solutions in rotor-stator and compressor cavities, as well as in-

gestion in rim seal complex geometries. The use of the proposed method in this area, including improved aero-therm-mechanical modelling, is expected to reduce uncertainty in clearance estimation and new efficient methodology in the design process. However, a full understanding of rim seal ingestion requires further detailed study of unsteady flow features.

Generalised circumferential Fourier transform for non-axisymmetric complex geometries, such as the novel method described in this thesis, is computationally attractive and accurate for different applications with temporal and spatial length-scale disparities. The method could be easily implemented in a standard time-domain CFD solver with a minimum influence on the flux calculation routine. It can contribute to a better understanding of the physics behind discrete flow disturbances representing by spatial harmonics, reducing the computational resources drastically. In summary, the results highlight, that the simulation of unsteady and non-axisymmetric flow phenomena such as instabilities in cavities, rim-seal ingestion, inlet distortion or rotating stall requires highly advanced time-domain techniques such as the Passage-Spectral Method to ensure an accurate prediction of the underlying interaction effects reducing the computational cost.

Future work

The studies performed are not exhaustive and a wider range of applications must be analysed to build a complete understanding of the advantages and disadvantages of the block-wise spatial Fourier model. The method's performance in multistage propagation distortion, for example, is yet to be assessed. A possible direction for future work could be the application of the proposed method to multi-row cases, developing a spatial Fourier method to capture instabilities and large-scale perturbation along different rows in a real engine, such in compressor stages. Further insights into distortion transfer modelling may be gained by studying the propagation of a sinusoidal disturbance through multiple stages taking advantage to the block Fourier reconstruction approach followed by the Passage-Spectral Method, where full-annulus geometries can be avoided and replace using the efficient proposed method which has shown to be accurate for capturing the interactions between different scales perturbations, due to the fact that the scattered modes are natively retained in the method as well. Since the method is able to repro-

duce interaction modes of different frequency waves, the application to rotor-stator numerical method could be directly assessed.

Another direction could be the study of the characterization of modes in the cases outlined in this research. In the last chapter of the thesis, it has been shown that rotating instabilities can happen under different conditions in various rotating cavities geometries. A stability analysis is required in order to identify and separate the driving parameters and to obtain a general description of the stability boundary for the different cavity geometries analysed. Furthermore, the non-linear stability analysis carried out for a generic fan configuration, can be applied to more representative and modern fan-stages, understanding also the stability behaviour of the different nodal diameters in such inlet distortion patterns and the potential effect of the downstream stator row, studying possible geometrical parameters and operating conditions which could affect to the stability limit. Since it is difficult to accurately control such large number of conditions in these unsteady simulations, theoretical or numerical methods are required to enhance the understanding of the rotating flow under these conditions, using the proposed Passage-Spectral Method as a efficient tool to carry out this investigation.

Bibliography

- [1] Stapelfeldt, S. C., 2013. “Advanced methods for multi-row forced response and flutter computations”. PhD thesis, Imperial College London.
- [2] Zhang, F., Wang, X., Li, J., and Zheng, D., 2017. “Computations on the flow characteristics and sealing performance of a stator well cavity for the second stage in gas turbine”. *Applied Thermal Engineering*, **125**, pp. 1300–1319.
- [3] Schouveiler, L., Le Gal, P., and Chauve, M. P., 2001. “Instabilities of the flow between a rotating and a stationary disk”. *Journal of Fluid Mechanics*, **443**, pp. 329–350.
- [4] Owen, J. M., Tang, H., and Lock, G. D., 2018. “Buoyancy-induced heat transfer inside compressor rotors: Overview of theoretical models”. *Aerospace*, **5**(1), p. 32.
- [5] Fidalgo, V. J., Hall, C., and Colin, Y., 2012. “A study of fan-distortion interaction within the nasa rotor 67 transonic stage”. *Journal of Turbomachinery*, **134**(5), p. 051011.
- [6] Hall, D. K., Greitzer, E. M., and Tan, C. S., 2017. “Analysis of fan stage conceptual design attributes for boundary layer ingestion”. *Journal of Turbomachinery*, **139**(7).
- [7] Loiodice, S., Tucker, P., and Watson, J., 2010. “Coupled open rotor engine intake simulations”. In Proceedings of the 48th AIAA Aerospace Sciences Meeting and Exhibit, pp. 2010–840.
- [8] Cao, T., Vadlamani, N. R., Tucker, P. G., Smith, A. R., Slaby, M., and Sheaf, C. T., 2017. “Fan–intake interaction under high incidence”. *Journal of Engineering for Gas Turbines and Power*, **139**(4), p. 041204.

-
- [9] Im, H.-S., Chen, X., and Zha, G.-C., 2010. “Detached eddy simulation of unsteady stall flows of a full annulus transonic rotor”. In ASME Turbo Expo 2010: Power for Land, Sea, and Air, American Society of Mechanical Engineers, pp. 2627–2642.
- [10] Lee, K.-B., Wilson, M., and Vahdati, M., 2017. “Numerical study on aeroelastic instability for a low-speed fan”. *Journal of Turbomachinery*, **139**(7), p. 071004.
- [11] Soderquist, D. R., Gorrell, S. E., and List, M. G., 2018. “Radial variation in distortion transfer and generation through a highly loaded fan stage”. In ASME Turbo Expo 2018: Turbomachinery Technical Conference and Exposition, American Society of Mechanical Engineers, pp. V02AT39A045–V02AT39A045.
- [12] Zhang, W., and Vahdati, M., 2018. “A parametric study of the effects of inlet distortion on fan aerodynamic stability”. *Journal of Turbomachinery*, **141**(1), p. 11 pages.
- [13] Day, I., and Cumpsty, N., 1978. “The measurement and interpretation of flow within rotating stall cells in axial compressors”. *Journal of Mechanical Engineering Science*, **20**(2), pp. 101–114.
- [14] Greitzer, E., 1980. “Axial compressor stall phenomena”. *Journal of Fluids Engineering*, **102**(2), pp. 134–151.
- [15] Tan, C., Day, I., Morris, S., and Wadia, A., 2010. “Spike-type compressor stall inception, detection, and control”. *Annual review of fluid mechanics*, **42**, pp. 275–300.
- [16] Choi, M., Smith, N. H., and Vahdati, M., 2013. “Validation of numerical simulation for rotating stall in a transonic fan”. *Journal of turbomachinery*, **135**(2), p. 021004.
- [17] Biesiadny, T. J., Braithwaite, W. M., Soeder, R. H., and Abdelwahab, M., 1986. “Summary of investigations of engine response to distorted inlet conditions”. *NASA Technical Memo No. 87317, Lewis Research Center, Cleveland, OH*.
- [18] Gunn, E. J., Tooze, S. E., Hall, C. A., and Colin, Y., 2013. “An experimental study of loss sources in a fan operating with continuous inlet stagnation pressure distortion”. *Journal of Turbomachinery*, **135**(5), p. 051002.

- [19] Stapelfeldt, S. C., and Vahdati, M., 2018. “Improving the flutter margin of an unstable fan blade”. *ASME Paper No. GT2018-76889*.
- [20] Camp, T., and Day, I., 1997. “A study of spike and modal stall inception indication in axial compressor”. *ASME Paper, 97-GT-526*.
- [21] Chen, J.-P., Hathaway, M. D., and Herrick, G. P., 2008. “Prestall behavior of a transonic axial compressor stage via time-accurate numerical simulation”. *Journal of Turbomachinery*, **130**(4), p. 041014.
- [22] Cao, C., Chew, J., Millington, P., and Hogg, S., 2004. “Interaction of rim seal and annulus flows in an axial flow turbine”. *Journal of engineering for gas turbines and power*, **126**(4), pp. 786–793.
- [23] Savov, S. S., Atkins, N. R., and Uchida, S., 2016. “Comparison of single and double lip rim seal geometry”. In *ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition*, American Society of Mechanical Engineers, pp. V05AT15A003–V05AT15A003.
- [24] Beard, P. F., Gao, F., Chana, K. S., and Chew, J., 2017. “Unsteady flow phenomena in turbine rim seals”. *Journal of Engineering for Gas Turbines and Power*, **139**(3).
- [25] Gao, F., Poujol, N., Chew, J. W., and Beard, P. F., 2018. “Advanced numerical simulation of turbine rim seal flows and consideration for rans turbulence modelling”. In *ASME Turbo Expo 2018: Turbomachinery Technical Conference and Exposition*, American Society of Mechanical Engineers Digital Collection.
- [26] Palermo, D. M., Gao, F., Chew, J. W., and Beard, P. F., 2019. “Effect of annulus flow conditions on turbine rim seal ingestion”. In *ASME Turbo Expo 2019: Turbomachinery Technical Conference and Exposition*, American Society of Mechanical Engineers Digital Collection.
- [27] Horwood, J., Hualca, F. P., Wilson, M., Scobie, J. A., Sangan, C. M., Lock, G. D., Dahlqvist, J., and Fridh, J., 2020. “Flow instabilities in gas turbine chute seals”. *Journal of Engineering for Gas Turbines and Power*, **142**(2).

-
- [28] Owen, J. M., 2010. “Thermodynamic analysis of buoyancy-induced flow in rotating cavities”. *Journal of Turbomachinery*, **132**(3), p. 031006.
- [29] Gentilhomme, O., Hills, N., Turner, A., and Chew, J., 2003. “Measurement and analysis of ingestion through a turbine rim seal”. *J. Turbomach.*, **125**(3), pp. 505–512.
- [30] Da Soghe, R., Bianchini, C., Sangan, C. M., Scobie, J. A., and Lock, G. D., 2017. “Numerical characterization of hot-gas ingestion through turbine rim seals”. *Journal of Engineering for Gas Turbines and Power*, **139**(3).
- [31] Hills, N. J., Chew, J. W., and Turner, A. B., 2002. “Computational and mathematical modeling of turbine rim seal ingestion”. *J. Turbomach.*, **124**(2), pp. 306–315.
- [32] Roy, R., Feng, J., Narzary, D., Saurabh, P., and Paolillo, R., 2004. “Experiments on gas ingestion through axial-flow turbine rim seals”. In *ASME Turbo Expo 2004: Power for Land, Sea, and Air*, American Society of Mechanical Engineers Digital Collection, pp. 347–358.
- [33] Savov, S. S., Atkins, N. R., and Uchida, S., 2017. “A comparison of single and double lip rim seal geometries”. *Journal of Engineering for Gas Turbines and Power*, **139**(11).
- [34] Schädler, R., Kalfas, A. I., Abhari, R. S., Schmid, G., and Voelker, S., 2017. “Modulation and radial migration of turbine hub cavity modes by the rim seal purge flow”. *Journal of Turbomachinery*, **139**(1).
- [35] Gourdain, N., Burguburu, S., Leboeuf, F., and Michon, G. J., 2010. “Simulation of rotating stall in a whole stage of an axial compressor”. *Computers & Fluids*, **39**(9), pp. 1644–1655.
- [36] Haug, J. P., and Niehuis, R., 2018. “Full annulus simulations of a transonic axial compressor stage with distorted inflow at transonic and subsonic blade tip speed”. *International Journal of Turbomachinery, Propulsion and Power*, **3**(1), p. 7.
- [37] He, L., 2010. “Fourier methods for turbomachinery applications”. *Progress in Aerospace Sciences*, **46**(8), pp. 329–341.

- [38] Dowell, E. H., and Hall, K. C., 2001. “Modeling of fluid-structure interaction”. *Annual review of fluid mechanics*, **33**(1), pp. 445–490.
- [39] Erdos, J., Alzner, E., and McNally, W., 1977. “Numerical solution of periodic transonic flow through a fan stage”. *AIAA journal*, **15**(11), pp. 1559–1568.
- [40] Giles, M. B., 1990. “Stator/rotor interaction in a transonic turbine”. *Journal of Propulsion and Power*, **6**(5), pp. 621–627.
- [41] Lindquist, D. R., and Giles, M. B., 1994. “Validity of linearized unsteady euler equations with shock capturing”. *AIAA journal*, **32**(1), pp. 46–53.
- [42] Ning, W., and He, L., 1998. “Computation of unsteady flows around oscillating blades using linear and nonlinear harmonic euler methods”. *Journal of Turbomachinery*, **120**(3), pp. 508–514.
- [43] Sbardella, L., and Imregun, M., 2001. “Linearized unsteady viscous turbomachinery flows using hybrid grids”. *J. Turbomach.*, **123**(3), pp. 568–582.
- [44] Corral, R., Escribano, A., Gisbert, F., Serrano, A., and Vasco, C., 2003. “Validation of a linear multigrid accelerated unstructured navier-stokes solver for the computation of turbine blades on hybrid grids”. *AIAA paper*, **3326**(9).
- [45] Hall, K. C., Thomas, J. P., and Clark, W. S., 2002. “Computation of unsteady nonlinear flows in cascades using a harmonic balance technique”. *AIAA journal*, **40**(5), pp. 879–886.
- [46] Crespo, J., Corral, R., and Pueblas, J., 2016. “An implicit harmonic balance method in graphics processing units for oscillating blades”. *Journal of Turbomachinery*, **138**(3), p. 031001.
- [47] Marshall, J., and Imregun, M., 1996. “A review of aeroelasticity methods with emphasis on turbomachinery applications”. *Journal of fluids and structures*, **10**(3), pp. 237–267.
- [48] Holmes, D. G., and Lorence, C. B., 1998. “Three dimensional linearized navier-stokes calculations for flutter and forced response”. In *Unsteady Aerodynamics and Aeroelasticity of Turbomachines*. Springer, pp. 211–224.

-
- [49] Clark, W. S., and Hall, K. C., 2000. “A time-linearized navier–stokes analysis of stall flutter”. *J. Turbomach.*, **122**(3), pp. 467–476.
- [50] Campobasso, M. S., and Giles, M. B., 2003. “Effects of flow instabilities on the linear analysis of turbomachinery aeroelasticity”. *Journal of propulsion and power*, **19**(2), pp. 250–259.
- [51] Giles, M. B., 1988. “Calculation of unsteady wake/rotor interaction”. *Journal of Propulsion and Power*, **4**(4), pp. 356–362.
- [52] Corral, R., Gallardo, J. M., and Vasco, C., 2007. “Aeroelastic stability of welded-in-pair low pressure turbine rotor blades: A comparative study using linear methods”.
- [53] Fernandez Aparicio, J. R., Serrano, A., and Vázquez, R., 2011. “On the linearity of turbomachinery interaction noise. part i: 2d analysis”. In 17th AIAA/CEAS Aeroacoustics Conference (32nd AIAA Aeroacoustics Conference), p. 2951.
- [54] Fernandez Aparicio, J. R., Serrano, A., and Vázquez, R. “On the linearity of turbomachinery interaction noise. part ii: 3d analysis”. In 18th AIAA/CEAS Aeroacoustics Conference (33rd AIAA Aeroacoustics Conference), p. 2309.
- [55] Serrano González, A., and Fernández Aparicio, J. R., 2015. “Turbine tone noise prediction using a linearized cfd solver: Comparison with measurements”. In Turbo Expo: Power for Land, Sea, and Air, Vol. 56642, American Society of Mechanical Engineers, p. V02BT41A004.
- [56] Gopinath, A., and Jameson, A., 2006. “Application of the time spectral method to periodic unsteady vortex shedding”. In 44th AIAA Aerospace Sciences Meeting and Exhibit, p. 449.
- [57] Wang, F., and di Mare, L., 2019. “Favre-averaged nonlinear harmonic method for compressible periodic flows”. *AIAA Journal*, **57**(3), pp. 1133–1142.
- [58] He, L., 1997. “Computational study of rotating-stall inception in axial compressors”. *Journal of Propulsion and Power*, **13**(1), pp. 31–38.

- [59] Frey, C., Ashcroft, G., Kersken, H.-P., and Voigt, C., 2014. “A harmonic balance technique for multistage turbomachinery applications”. In *Turbo Expo: Power for Land, Sea, and Air*, Vol. 45615, American Society of Mechanical Engineers, p. V02BT39A005.
- [60] Sotillo, A., and Gallardo, J. M., 2018. “Study of the impact of multi-row interaction on flutter analysis for a representative lpt geometry”. In *ASME Turbo Expo 2018: Turbomachinery Technical Conference and Exposition*, American Society of Mechanical Engineers Digital Collection.
- [61] He, L., 2006. “Fourier modeling of steady and unsteady nonaxisymmetrical flows”. *Journal of propulsion and power*, **22**(1), pp. 197–201.
- [62] He, L., 2011. “Efficient computational model for nonaxisymmetric flow and heat transfer in rotating cavity”. *Journal of Turbomachinery*, **133**(2), 10. 021018.
- [63] He, L., 2011. “Block-spectral approach to film-cooling modeling”. *Journal of turbomachinery*, **134**(2), 06. 021018.
- [64] He, L., 2013. “Block-spectral mapping for multi-scale solution”. *Journal of Computational Physics*, **250**, pp. 13–26.
- [65] He, L., 2013. “Fourier spectral modelling for multi-scale aero-thermal analysis”. *International Journal of computational fluid dynamics*, **27**(2), pp. 118–129.
- [66] He, L., 2018. “Multiscale block spectral solution for unsteady flows”. *International Journal for Numerical Methods in Fluids*, **86**(10), pp. 655–678.
- [67] Stapelfeldt, S. C., and Di Mare, L., 2015. “Reduced passage method for multirow forced response computations”. *AIAA Journal*, **53**(10), pp. 3049–3062.
- [68] Tyler, J. M., and Sofrin, T. G., 1962. Axial flow compressor noise studies. Tech. rep., SAE Technical Paper.
- [69] Corral, R., June 1990. “Resolución de las ecuaciones de euler en régimen transónico mediante métodos multimalla.”. PhD thesis, Escuela Técnica Superior de Ingenieros Aeronáuticos, Universidad Politécnica de Madrid.

-
- [70] Gómez, R., 2000. “Una estructura de datos basada en aristas para la resolución de las ecuaciones de navier-stokes”. PhD thesis, Escuela Técnica Superior de Ingenieros Aeronáuticos, Universidad Politécnica de Madrid, Madrid, Spain.
- [71] Contreras, J., and Corral, R., May 2000. “Quantitative influence of the steady non-reflective boundary conditions on blade-to-blade computations.”. In ASME Paper 2000-GT-0515. 45th ASME Gas Turbine and Aeroengine Congress, Munich.
- [72] Corral, R., and neda, J. F.-C., January 2001. “Surface mesh generation by means of steiner triangulations.”. *AIAA Journal*, **39**(1), pp. 176–180.
- [73] Contreras, J., Corral, R., neda, J. F.-C., Pastor, G., and Vasco, C., 2002. “Semiunstructured grid methods for turbomachinery applications.”. In ASME paper 2002-GT-30572.
- [74] Gisbert, F., Corral, R., and Pueblas, J., 2013. “Computation of turbomachinery flows with a parallel unstructured mesh navier-stokes equations solver on gpus”. In 21st AIAA Computational Fluid Dynamics Conference, p. 2864.
- [75] Pueblas, J., 2017. “Acceleration of an unstructured compressible flow solver using implicit methods and graphics cards”. PhD thesis, Escuela Técnica Superior de Ingenieros Aeronáuticos, Universidad Politécnica de Madrid, Madrid, Spain.
- [76] Burgos, M. A., Chia, J. M., Corral, R., and López, C., 2010. “Rapid meshing of turbomachinery rows using semi-unstructured multi-block conformal grids”. *Engineering with Computers*, **26**(4), pp. 351–362.
- [77] Gisbert, F., Corral, R., and Pastor, G., 2011. “Implementation of an edge-based navier-stokes solver for unstructured grids in graphics processing units”. *ASME Paper No. GT2011-46224*.
- [78] Lohner, R., 1993. “Matching semi-structured and structured grids for navier-stokes calculations.”. In AIAA paper 93-3348.
- [79] Jameson, A., 1991. “Time dependent calculations using multigrid, with applications to unsteady flows past airfoils and wings”. In 10th Computational Fluid Dynamics Conference, p. 1596.

- [80] Zhang, W., and Vahdati, M., 2017. “Influence of the inlet distortion on fan stall margin at different rotational speed”. In GPPS conference paper, Vol. 207.
- [81] Lee, K.-B., Wilson, M., and Vahdati, M., 2019. “Effects of inlet disturbances on fan stability”. *Journal of Engineering for Gas Turbines and Power*, **141**(5), p. 051014.
- [82] Jiang, H., Lu, Y., Yuan, W., and Li, Q., 2009. “Experimental investigation of the influence of inlet distortion on the stall inception in a low speed axial compressor”. In ASME turbo expo 2009: Power for land, sea, and air, American Society of Mechanical Engineers Digital Collection, pp. 1533–1542.
- [83] Brailko, I., Milesin, V., Volkov, A., and Korznev, V., 2010. “Numerical and experimental investigations of crf with simulation of flow non-uniformity in the basic flight conditions”. In 27th Congress of International Council of the Aeronautical Sciences, ICAS, Vol. 4.
- [84] Lesser, A., Iseler, J., and Niehuis, R., 2011. “Numerical investigation of a highly loaded axial compressor stage with inlet distortions”. In ASME 2011 Turbo Expo: Turbine Technical Conference and Exposition, American Society of Mechanical Engineers Digital Collection, pp. 381–391.
- [85] Sharma, G., Zori, L., Connell, S., and Godin, P., 2013. “Efficient computation of large pitch ratio transonic flow in a fan with inlet distortion”. In ASME Turbo Expo 2013: Turbine Technical Conference and Exposition, American Society of Mechanical Engineers, pp. V06CT42A025–V06CT42A025.
- [86] Lobo, B., Zori, L., Galpin, P., and Holmes, W., 2016. “Efficient modeling strategy of an axial compressor fan-stage under inlet distortion”. In ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition, American Society of Mechanical Engineers Digital Collection.
- [87] Biesinger, T., Cornelius, C., Rube, C., Braune, A., Campregher, R., Godin, P. G., Schmid, G., and Zori, L., 2010. “Unsteady cfd methods in a commercial solver for turbomachinery applications”. *Turbo Expo: Power for Land, Sea, and Air*, **7**, 06, pp. 2441–2452.

- [88] Li, H., and He, L., 2002. "Single-passage analysis of unsteady flows around vibrating blades of a transonic fan under inlet distortion". *Journal of Turbomachinery*, **124**(2), pp. 285–292.
- [89] Carnevale, M., Wang, F., Parry, A. B., Green, J. S., and di Mare, L., 2018. "Fan similarity model for the fan-intake interaction problem". *Journal of Engineering for Gas Turbines and Power*, **140**(5), p. 051202.
- [90] Inoue, M., Kuroumaru, M., Tanino, T., Yoshida, S., and Furukawa, M., 2001. "Comparative studies on short and long length-scale stall cell propagating in an axial compressor rotor". *Journal of Turbomachinery*, **123**(1), pp. 24–30.
- [91] Hah, C., Bergner, J., and Schiffer, H.-P., 2006. "Short length-scale rotating stall inception in a transonic axial compressor: criteria and mechanisms". In *ASME turbo expo 2006: Power for land, sea, and air*, American Society of Mechanical Engineers, pp. 61–70.
- [92] Lee, K.-B., Dodds, J., Wilson, M., and Vahdati, M., 2018. "Validation of a numerical model for predicting stalled flows in a low-speed fan - part ii: Unsteady analysis". *Journal of Turbomachinery*, **140**(5), p. 051009.
- [93] Cousins, W. T., 2004. "History, philosophy, physics, and future directions of aircraft propulsion system/inlet integration". In *ASME Turbo Expo 2004: Power for Land, Sea, and Air*, American Society of Mechanical Engineers Digital Collection, pp. 305–320.
- [94] Burstadt, P. L., Calogeras, J., and Mehalic, C., 1971. "Experimental investigation of the effect of screen induced total pressure distortion on turbojet stall margin". *Technical Memorandum NASA TM X-2239, NASA Lewis Research Center, Cleveland, OH*.
- [95] Rademakers, R., Bindl, S., Niehuis, R., et al., 2016. "Effects of flow distortions as they occur in s-duct inlets on the performance and stability of a jet engine". *Journal of Engineering for Gas Turbines and Power*, **138**(2).
- [96] Page, J. H., Hield, P., and Tucker, P. G., 2018. "Effect of inlet distortion features on transonic fan rotor stall". *Journal of Turbomachinery*, **140**(7), p. 071008.
- [97] Schneck, W., Ferrar, A., Bailey, J., Hoopes, K., and O'Brien, W., 2013. "Improved pre-

- diction method for the design of high-resolution total pressure distortion screens”. In 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, p. 1133.
- [98] Hah, C., Rabe, D. C., Sullivan, T. J., and Wadia, A. R., 1998. “Effects of inlet distortion on the flow field in a transonic compressor rotor”. *Journal of turbomachinery*, **120**(2), pp. 233–246.
- [99] Ferrar, A., Schneck, W., Bailey, J., Hoopes, K., and O’Brien, W., 2012. “Application of additive manufacturing to rapidly produce high-resolution total pressure distortion screens”. In 50th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, p. 929.
- [100] Reid, C., 1969. “The response of axial flow compressors to intake flow distortion”. In ASME 1969 Gas Turbine Conference and Products Show, American Society of Mechanical Engineers, pp. V001T01A029–V001T01A029.
- [101] Longley, J., Shin, H.-W., Plumley, R., Silkowski, P., Day, I., Greitzer, E., Tan, C., and Wisler, D., 1996. “Effects of rotating inlet distortion on multistage compressor stability”. *Journal of Turbomachinery*, **118**(2), pp. 181–188.
- [102] Perovic, D., Hall, C. A., and Gunn, E., 2019. “Stall inception in a boundary layer ingesting fan”. *Journal of Turbomachinery*, **141**(9), p. 091007.
- [103] Yao, J., Gorrell, S. E., and Wadia, A. R., 2010. “High-fidelity numerical analysis of per-rev-type inlet distortion transfer in multistage fans - part ii: Entire component simulation and investigation”. *Journal of Turbomachinery*, **132**(4), p. 041015.
- [104] Choi, M., Vahdati, M., and Imregun, M., 2011. “Effects of fan speed on rotating stall inception and recovery”. *Journal of Turbomachinery*, **133**(4), p. 041013.
- [105] Sheoran, Y., Bouldin, B., and Krishnan, P. M., 2012. “Compressor performance and operability in swirl distortion”. *Journal of turbomachinery*, **134**(4).
- [106] Hall, K. C., and Lorence, C. B., 1992. “Calculation of three-dimensional unsteady flows in turbomachinery using the linearized harmonic euler equations”. In ASME 1992 In-

- ternational Gas Turbine and Aeroengine Congress and Exposition, American Society of Mechanical Engineers, pp. V005T14A013–V005T14A013.
- [107] Burgos, M. A., Contreras, J., and Corral, R., 2011. “Efficient edge-based rotor/stator interaction method”. *AIAA journal*, **49**(1), pp. 19–31.
- [108] Corral, R., Gisbert, F., and Pueblas, J., 2017. “Efficient execution of a parallel edged-based navier-stokes solver on graphics processing units”. *Int. J. Comp. Fluid Dynamics*, **31**(2), pp. 1–16.
- [109] Peterson, M. W., Gorrell, S. E., and List, M. G., 2017. “Fourier descriptors for improved analysis of distortion transfer and generation”. In ASME Turbo Expo 2017: Turbomachinery Technical Conference and Exposition, American Society of Mechanical Engineers Digital Collection.
- [110] Kodama, H., 1985. “Performance of axial compressor with non-uniform exit static pressure”. In ASME 1985 Beijing International Gas Turbine Symposium and Exposition, American Society of Mechanical Engineers Digital Collection.
- [111] Rife, M. E., Barbarossa, F., Di Mare, L., Parry, A. B., and Green, J. S., 2018. “Radical guide vane design and optimization”. *Journal of Propulsion and Power*, **34**(6), pp. 1455–1466.
- [112] Di Mare, L., Carnevale, M., Rife, M. E., Kulkarni, D., and Northall, R., 2019. “Modelling and design of lpc components with semi-analytical models: Part ii - design of optimal ogv cascades”. In Turbo Expo: Power for Land, Sea, and Air, Vol. 58561, American Society of Mechanical Engineers, p. V02BT42A003.
- [113] Enoki, T., Kodama, H., and Kusuda, S., 2013. “Investigation of fan rotor interaction with pressure disturbance produced by downstream pylon”. In ASME Turbo Expo 2013: Turbine Technical Conference and Exposition, American Society of Mechanical Engineers Digital Collection.
- [114] Shahpar, S., Giacche, D., and Lapworth, L., 2003. “Multi-objective design and optimisation of bypass outlet-guide vanes”. In ASME Turbo Expo 2003, collocated with the

- 2003 International Joint Power Generation Conference, American Society of Mechanical Engineers Digital Collection, pp. 591–601.
- [115] Oishi, T., Kusuda, S., Kodama, H., and Namba, M., 2012. “Buzz saw noise behavior due to influence of potential perturbation”. In 18th AIAA/CEAS Aeroacoustics Conference (33rd AIAA Aeroacoustics Conference), p. 2266.
- [116] Tschirner, T., Pfitzner, M., Merz, R., et al., 2002. “Aerodynamic optimisation of an aeroengine bypass duct ogv-pylon configuration”. In ASME Turbo Expo 2002: Power for Land, Sea, and Air, American Society of Mechanical Engineers Digital Collection, pp. 879–888.
- [117] Xie, Y., Yang, X., Ding, J., Liu, X., and Feng, J., 2014. “The optimization of an integrated strut ogv configuration”. In ASME Turbo Expo 2014: Turbine Technical Conference and Exposition, American Society of Mechanical Engineers Digital Collection.
- [118] Goulos, I., MacManus, D., and Sheaf, C., 2019. “Civil turbofan engine exhaust aerodynamics: Impact of fan exit flow characteristics”. *Aerospace Science and Technology*, **93**, p. 105181.
- [119] Barbarossa, F., Rife, M., Carnevale, M., Parry, A., Green, J., and Di Mare, L., 2017. “Fast optimisation of a three-dimensional bypass system using a new aerodynamic design method”. In ASME Turbo Expo 2017: Turbomachinery Technical Conference and Exposition, American Society of Mechanical Engineers Digital Collection.
- [120] Boudet, J., Autef, V., Chew, J., Hills, N., and Gentilhomme, O., 2005. “Numerical simulation of rim seal flows in axial turbines”. *The Aeronautical Journal*, **109**(1098), pp. 373–383.
- [121] Gao, F., Chew, J., Beard, P. F., Amirante, D., and Hills, N., 2017. “Numerical studies of turbine rim sealing flows on a chute seal configuration”. In Proceedings of 12th European conference on turbomachinery fluid dynamics & thermodynamics.
- [122] Berg, R. A., Tan, C., Ding, Z., Laskowski, G., Palafox, P., and Miorini, R., 2018. “Exper-

- imental and analytical assessment of cavity modes in a gas turbine wheelspace”. *Journal of Engineering for Gas Turbines and Power*, **140**(6), p. 062502.
- [123] Hualca, F. P., Horwood, J., Sangan, C. M., Lock, G. D., and Scobie, J. A., 2020. “The effect of vanes and blades on ingress in gas turbines”. *Journal of Engineering for Gas Turbines and Power*, **142**(2).
- [124] Abe, T., Kikuchi, J., and Takeuchi, H., 1979. “An investigation of turbine disc cooling”. In Paper GT30, 3rd CIMAC Congress, Vienna.
- [125] Phadke, U., and Owen, J., 1988. “Aerodynamic aspects of the sealing of gas-turbine rotor-stator systems: Part 3: The effect of nonaxisymmetric external flow on seal performance”. *International Journal of Heat and Fluid Flow*, **9**(2), pp. 113–117.
- [126] Johnson, B., Mack, G., Paolillo, R., and Daniels, W., 1994. “Turbine rim seal gas path flow ingestion mechanisms”. *AIAA paper*, **94**, p. 2703.
- [127] Daily, J. W., and Nece, R. E., 1960. “Chamber dimension effects on induced flow and frictional resistance of enclosed rotating disks”. *Journal of basic engineering*, **82**(1), pp. 217–230.
- [128] Chew, J. W., and Vaughan, C. M., 1988. “Numerical predictions for the flow induced by an enclosed rotating disc”. In ASME 1988 International Gas Turbine and Aeroengine Congress and Exposition, American Society of Mechanical Engineers Digital Collection.
- [129] Lopez, J. M., Hart, J. E., Marques, F., Kittelman, S., and Shen, J., 2002. “Instability and mode interactions in a differentially driven rotating cylinder”. *Journal of Fluid Mechanics*, **462**, pp. 383–409.
- [130] Lopez, J., 2006. “Rotating and modulated rotating waves in transitions of an enclosed swirling flow”. *Journal of Fluid Mechanics*, **553**, pp. 323–346.
- [131] Launder, B., Poncet, S., and Serre, E., 2010. “Laminar, transitional, and turbulent flows in rotor-stator cavities”. *Annual review of fluid mechanics*, **42**, pp. 229–248.
- [132] Owen, J. M., and Rogers, R. H., 1989. “Flow and heat transfer in rotating-disc systems”. *vol. 1, Research Studies Press, Taunton, UK and John Wiley, NY.*

- [133] von Karman, T., 1921. “Laminar and turbulence reibung”. *Z. Angew. Math. Mech*, **1**, pp. 233–244.
- [134] Batchelor, G. K., 1951. “Note on a class of solutions of the Navier-Stokes equations representing steady rotationally-symmetric flow”. *Quart. J. Mech. Appl. Math*, **IV**, pt. **1**, pp. 29–41.
- [135] Dijkstra, D., and Van Heijst, G. J. F., 1983. “The flow between two finite rotating disks enclosed by a cylinder”. *Journal of Fluid Mechanics*, **128**, pp. 123–154.
- [136] Brady, J. F., and Durlofsky, L., 1987. “On rotating disk flow”. *Journal of Fluid Mechanics*, **175**, pp. 363–394.
- [137] Itoh, M., Yamada, Y., Imao, S., and Gonda, M., 1992. “Experiments on turbulent flow due to an enclosed rotating disk”. *Experimental thermal and fluid science*, **5**(3), pp. 359–368.
- [138] Czarny, O., Iacovides, H., and Launder, B. E., 2002. “Precessing vortex structures in turbulent flow within rotor-stator disc cavities”. *Flow, Turbulence and Combustion*, **69**(1), pp. 51–61.
- [139] Bohn, D., Emunds, R., Gorzelitz, V., and Krüger, U., 1996. “Experimental and theoretical investigations of heat transfer in closed gas-filled rotating annuli 2”. *Journal of turbomachinery*, **118**(CONF-940626-).
- [140] Bohn, D., Ren, J., and Tuemmers, C. “Investigation of the unstable flow structure in a rotating cavity”. In *ASME Turbo Expo 2006: Power for Land, Sea, and Air*, American Society of Mechanical Engineers Digital Collection, pp. 1433–1442.
- [141] SUN, Z., LINDBLAD, K., CHEW, J. W., and YOUNG, C., 2007. “Les and rans investigations into buoyancy-affected convection in a rotating cavity with a central axial throughflow”. *Journal of engineering for gas turbines and power*, **129**(2), pp. 318–325.
- [142] Chew, J. W., Gao, F., and Palermo, D. M., 2018. “Flow mechanisms in axial turbine rim sealing”. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, p. 0954406218784612.

- [143] Horwood, J., Hualca, F. P., Scobie, J. A., Wilson, M., Sangan, C. M., and Lock, G. D., 2019. “Experimental and computational investigation of flow instabilities in turbine rim seals”. *Journal of Engineering for Gas Turbines and Power*, **141**(1).
- [144] Jakoby, R., Zierer, T., Lindblad, K., Larsson, J., Bohn, D. E., Funcke, J., Decker, A., et al., 2004. “Numerical simulation of the unsteady flow field in an axial gas turbine rim seal configuration”. In *ASME Turbo Expo 2004: Power for Land, Sea, and Air*, American Society of Mechanical Engineers, pp. 431–440.
- [145] Zhou, D., Roy, R., Wang, C.-Z., and Glahn, J., 2011. “Main gas ingestion in a turbine stage for three rim cavity configurations”. *Journal of turbomachinery*, **133**(3).
- [146] Milne-Thomson, L. M., Abramowitz, M., and Stegun, I., 1972. “Handbook of mathematical functions”. *Handbook of Mathematical Functions*.
- [147] Van Eeten, K. M. P., Van der Schaaf, J., Schouten, J. C., and Van Heijst, G. J. F., 2012. “Boundary layer development in the flow field between a rotating and a stationary disk”. *Physics of Fluids*, **24**(3), p. 033601.
- [148] Bödewadt, U., 1940. “Die drehströmung über festem grunde”. *ZAMM-Journal of Applied Mathematics and Mechanics/Zeitschrift für Angewandte Mathematik und Mechanik*, **20**(5), pp. 241–253.
- [149] Gärtner, W., 1998. “A momentum integral method to predict the frictional torque of a rotating disk with protruding bolts”. In *ASME 1998 International Gas Turbine and Aeroengine Congress and Exhibition*, pp. V004T09A029–V004T09A029.
- [150] Chew, J. W., and Hills, N. J., 2013. “Rotating flow and heat transfer in cylindrical cavities with radial inflow”. *Journal of Engineering for Gas Turbines and Power*, **135**, pp. 032502–1.
- [151] Sun, Z., Amirante, D., Chew, J. W., and Hills, N. J., 2016. “Coupled aerothermal modeling of a rotating cavity with radial inflow”. *Journal of Engineering for Gas Turbines and Power*, **138**(3), p. 032505.

- [152] Owen, J. M., Pincombe, J. R., and Rogers, R. H., 1985. “Source–sink flow inside a rotating cylindrical cavity”. *Journal of Fluid Mechanics*, **155**, pp. 233–265.
- [153] van Eeten, K. M. P., van der Schaaf, J., van Heijst, G. J. F., and Schouten, J. C., 2013. “Lyapunov-stability of solution branches of rotating disk flow”. *Physics of Fluids*, **25**(7), p. 073602.
- [154] Lopez, J. M., Marques, F., Rubio, A. M., and Avila, M., 2009. “Crossflow instability of finite bödewadt flows: Transients and spiral waves”. *Physics of Fluids*, **21**(11), p. 114107.
- [155] Ganine, V., Javiya, U., Hills, N., and Chew, J., 2012. “Coupled fluid-structure transient thermal analysis of a gas turbine internal air system with multiple cavities”. *Journal of Engineering for Gas turbines and Power*, **134**(10), p. 102508.
- [156] Altuna, A., Chaquet, J. M., Corral, R., Gisbert, F., and Pastor, G., 2013. “Application of a fast loosely coupled fluid/solid heat transfer method to the transient analysis of low-pressure-turbine disk cavities”. In ASME Turbo Expo 2013: Turbine Technical Conference and Exposition, pp. V03BT11A017–V03BT11A017.
- [157] Lopez, J. M., and Weidman, P. D., 1996. “Stability of stationary endwall boundary layers during spin-down”. *Journal of Fluid Mechanics*, **326**, pp. 373–398.

A. Harmonic Content of Non-uniform Mean Flow Velocity Wave Equation

The encoding of information in a carrier wave by varying the instantaneous frequency of the wave is known as Frequency Modulation (FM). This appendix sets out the basic properties of modulated signals. In the context of FM, the important thing is to realize that instantaneous frequency follows naturally from the fact that the frequency of the carrier varies continuously in response to the modulating wave (i.e., the baseband signal). If the information to be transmitted in time is $x(t) = \sin[\theta(t)]$ then

$$\theta(t) = \int^t \omega(t') dt' \quad (\text{A.1})$$

being

$$\omega(t) = \omega_L + \varepsilon \omega \cos(\omega_c t + \phi) \quad (\text{A.2})$$

where ω_L is the carrier's base frequency and ω_c corresponds to the modulated frequency or baseband signal. Integrating Eqn. (A.1) having into account Eqn. (A.2) we can obtain the solution for $\theta(t)$

$$\theta(t) = \omega_L t + \beta \sin(\omega_c t + \phi) \quad (\text{A.3})$$

where $\beta = \varepsilon \frac{\omega_L}{\omega_c}$, which indicates by how much the modulated variable varies around its unmodulated level. It relates to variations in the carrier frequency. The carrier frequency, ω_L , is then the highest frequency component present in the modulating signal $x(t)$, and ε is a factor to control the maximum deviation of the instantaneous frequency from the carrier frequency. For a sine wave modulation, the modulation index β is seen to be the ratio of the peak frequency

deviation of the carrier wave to the frequency of the modulating sine wave.

By manipulating the mathematics, and transforming the frequency domain in spatial harmonics, now we solve the signal $u(x, t)$ instead of $x(t)$. Transforming into space we have to take into account that $\omega_L t = k_L(x - c_0 t)$, being c_0 the phase speed, where k_L is the carrier spatial harmonic of the signal. In that way $\omega_c t = k_c x$, being k_c the modulated harmonic. A general formula for $u(x, t)$ of one sinusoid by another can be written then for $\phi = 0$ as

$$u(x, t) = u_0 \sin[k_L(x - c_0 t) + \beta \sin(k_c t)] \quad (\text{A.4})$$

where now $\beta = \varepsilon \frac{k_L}{k_c}$ and u_0 is the carrier's harmonic amplitude. Expanding Eqn. (A.4):

$$u(x, t) = u_0 \{ \sin[k_L(x - c_0 t)] \cos[\beta \sin(k_c t)] + \cos[k_L(x - c_0 t)] \sin[\beta \sin(k_c t)] \} \quad (\text{A.5})$$

The harmonic distribution of a sine wave carrier modulated by such a sinusoidal signal can be represented with Bessel functions; this provides the basis for a mathematical understanding of harmonics modulation. The modulation of any carrier in any way produces sidebands. For the case of a carrier modulated by a single sine wave, the resulting harmonic spectrum can be calculated using Bessel functions of the first kind, as a function of the sideband number and the modulation index. Using the properties of the Bessel function such as

$$\begin{aligned} \cos[\beta \sin(k_c t)] &= J_0(\beta) + 2[J_2(\beta) \cos(2k_c t) + J_4(\beta) \cos(4k_c t) + \dots] \\ \sin[\beta \sin(k_c t)] &= +2[J_1(\beta) \sin(k_c t) + J_3(\beta) \sin(3k_c t) + \dots] \end{aligned} \quad (\text{A.6})$$

It is possible to solve the basic Bessel function equation and express Eqn. (A.5) in the format:

$$\begin{aligned} \frac{u(x, t)}{u_0} &= J_0(\beta) \sin[k_L(x - c_0 t)] + \\ &J_1(\beta) \{ \cos[(k_c - k_L)x - k_L c_0 t] - \cos[(k_c + k_L)x - k_L c_0 t] \} + \\ &J_2(\beta) \{ \cos[(2k_c - k_L)x - k_L c_0 t] - \cos[(2k_c + k_L)x - k_L c_0 t] \} + \\ &\dots \\ &J_n(\beta) \{ \cos[(nk_c - k_L)x - k_L c_0 t] - \cos[(nk_c + k_L)x - k_L c_0 t] \} \end{aligned} \quad (\text{A.7})$$

Figure A.1.: Harmonic content for $k_L = 1$ and $k_c = 6$.

In this form, it can be seen in Eqn. (A.7) that the carrier spatial harmonic term, k_L , now varies in two sideband components $k_c - k_L$ and $k_c + k_L$, where k_c is now the modulated harmonic. A low-harmonic spectrum is translated to a band that extends above and below the carrier harmonic. It is the spectrum of the integral of the baseband signal that appears in the band surrounding the carrier harmonic. The modulated sidebands are dependent on both the level of deviation and the harmonic of the modulation. In fact the total spectrum for a harmonic modulated signal consists of the carrier plus an infinite number of sidebands spreading out on either side of the carrier at integral multiples of the modulating harmonic. The mathematical representation of harmonic modulation consists of a sinusoidal expression with the integral of the baseband signal added to the argument of the sine or cosine function.

Table A.1 shows the values of harmonic modulation side in Fig. A.1 for different modulation index β . This demonstrates very clearly that the modulation index influences the harmonic content of the modulated waveform. The modulation index can be used to make the harmonic deviation more sensitive or less sensitive to variations in the baseband value. For small values of modulation index, when using narrow-band modulation, the signal consists of the carrier and the two sidebands spaced at the modulated harmonic either side of the carrier. Having

β	$J_0(\beta)$	$J_1(\beta)$	$J_2(\beta)$	$J_3(\beta)$	$J_4(\beta)$	$J_5(\beta)$	$J_6(\beta)$
0.5	0.94	0.24	0.03	0.00	0.00	0.00	0.00
1	0.76	0.44	0.11	0.02	0.00	0.00	0.00
2	0.22	0.58	0.35	0.13	0.03	0.00	0.00
10	0.25	0.04	0.25	0.06	0.22	0.23	0.01

Table A.1.: Bessel functions values of the first kind $J_n(\beta)$ for modulation index cases in A.1.

into account the value of first kind Bessel functions, $J_n(\beta)$, in the particular case of $\beta \ll 1$ it is well known that $J_0(\beta) \simeq 1$ and $J_1(\beta) \simeq \frac{\beta}{2}$. The sidebands further out are minimal and can be ignored. As the level of the modulation index is increased other sidebands at twice the modulation harmonic start to appear. Further increases in modulation index result in the level of other sidebands increasing in level. An interesting property of the solution shown in Fig. A.1 and the values of the Bessel function in Tab. A.1 is given by Neumann's addition theorem [146], where

$$1 = J_0(0) = J_0^2(\beta) + 2 \sum_{n=1}^{\infty} J_n^2(\beta) \quad (\text{A.8})$$

This gives rise to the conclusion that the the harmonic distribution of a sine wave carrier modulated by a sinusoidal signal does not contribute to modify its total energy and the system is in equilibrium by adding sideband contribution surrounding the carrier harmonic.

B. Modelling of Unsteady Fluid Flow and Heat Transfer in a Rotor-Stator Cavity

Contents

B.1. One-Dimensional Analytical Model	184
B.2. Comparison with two-dimensional axisymmetric simulations	210
B.3. Summary	217

The unsteady laminar flow between two large rotating disks when one of them is impulsively started is described using the von Kármán similarity analysis to reduce the solution of the Navier-Stokes equations to a set of ordinary differential equations. The appendix is structured as follows: firstly, the system of ODEs governing the unsteady viscous flow between two large rotating disks when one of them is impulsively started is described. Secondly, the dedicated numerical method used to solve this set of ODEs is outlined. Thirdly, the method is used to obtain solutions at different Reynolds numbers and conclusions are drawn for the evolution of the velocity and temperature fields. Numerical simulations are supported with a theoretical analysis to ease the understanding of the underlying physics. Finally, two-dimensional laminar simulations for the transient velocity and temperature field in a rotor-stator cavity with a cylindrical housing are presented, and the results are compared with the self-similar solutions. It is confirmed that the two solutions compare well in the innermost region of the cavity and that the

quasi-steady approximation of the fluid is only valid if the characteristic time of the transient $\Omega t_c \gg (\Omega s^2/\nu)^{1/2}$, which is a relevant conclusion for the computation of complex gas turbine cavity analyses.

B.1. One-Dimensional Analytical Model

B.1.1. Governing Equations

Let us consider two infinite coaxial disks separated by a spacing s (see Fig. B.1). A first disk, named rotor, rotates with angular velocity Ω , whereas a second, named stator, is stationary. The wall temperatures of the rotor and the stator, T_R and T_S , respectively, are supposed to be uniform. The fluid has constant density, ρ , and kinematic viscosity, ν , and the flow is assumed to be laminar and axisymmetric.

B.1.1.1. Velocity Field

The flow pattern consists of two separated boundary-layers attached to both disks if the Reynolds number is large enough (see Fig. B.2). This allows simplification of the full Navier-Stokes equations using the boundary-layer approximation, where it is assumed that the flow variations in the radial direction are much smaller than in the wall-normal direction. This can be expressed in mathematical form as $\partial_r \ll \partial_z$. The thin form of the continuity and momentum equations for incompressible and axisymmetric flow in cylindrical coordinates (r, θ, z) , where t is the physical

Figure B.1.: Physical model of two coaxial rotating infinite disks.

time, p is the pressure and $\mathbf{v}$ is the velocity, is:

$$\begin{aligned}
 0 &= \frac{\partial v_r}{\partial r} + \frac{v_r}{r} + \frac{\partial v_z}{\partial z} & [a] \\
 \frac{\partial v_r}{\partial t} &= -v_r \frac{\partial v_r}{\partial r} - v_z \frac{\partial v_r}{\partial z} + \frac{v_\theta^2}{r} - \frac{1}{\rho} \frac{\partial p}{\partial r} + \mathbf{v} \left(\nabla^2 v_r - \frac{v_r}{r^2} \right) & [b] \\
 \frac{\partial v_\theta}{\partial t} &= -v_r \frac{\partial v_\theta}{\partial r} - v_z \frac{\partial v_\theta}{\partial z} - \frac{v_r v_\theta}{r} + \mathbf{v} \left(\nabla^2 v_\theta - \frac{v_\theta}{r^2} \right) & [c] \\
 \frac{\partial v_z}{\partial t} &= -v_r \frac{\partial v_z}{\partial r} - v_z \frac{\partial v_z}{\partial z} - \frac{1}{\rho} \frac{\partial p}{\partial z} + \mathbf{v} \nabla^2 v_z & [d]
 \end{aligned} \tag{B.1}$$

where

$$\nabla^2 = \frac{\partial^2}{\partial r^2} + \frac{1}{r} \frac{\partial}{\partial r} + \frac{\partial^2}{\partial z^2} \tag{B.2}$$

A similarity solution was found by [133] for an infinite rotating disk where the flow is at rest. By using this technique he managed to reduce the full Navier-Stokes equations to three non-linear Ordinary Differential Equations (ODEs). The similarity principle also applies if the fluid far away from the wall rotates as a rigid body [134] and even in the case of two infinite coaxial rotating disks, which is actually our case.

The velocity vector in a cylindrical system of coordinates in the absolute frame of reference can be written as, $\mathbf{v} = v_r \mathbf{e}_r + v_\theta \mathbf{e}_\theta + v_z \mathbf{e}_z$, where $\mathbf{e}_r$, $\mathbf{e}_\theta$, and $\mathbf{e}_z$, are unit vectors along the radial, circumferential and axial directions, respectively. Different to previous works [133, 134], we allow variations of the velocity with time. According to well-established nomenclature, the three components of the velocity can be expressed as

$$v_r = \Omega r F(\xi, \tau); \quad v_\theta = \Omega r G(\xi, \tau); \quad v_z = (\Omega \nu)^{1/2} H(\xi, \tau) \tag{B.3}$$

where F , G and H are, respectively, the dimensionless radial, circumferential and axial components of the velocity, and

$$\xi = \frac{z}{s} \quad \text{and} \quad \tau = \Omega t \tag{B.4}$$

are the dimensionless axial length and time. It is noteworthy that the characteristic velocity in the axial direction, $v_{zc} \sim (\Omega \nu)^{1/2}$ is in general much smaller than the characteristic velocity in

Figure B.2.: Sketch of the flow pattern between two infinite disks.

the radial and circumferential directions, Ωr . The ratio between both is $v_{\theta c}/v_{zc} \sim r_c/\delta_c$, where $\delta_c \sim (\nu/\Omega)^{1/2}$ is the characteristic thickness of the disks boundary-layers. The dimensionless pressure, $P(\xi)$, is defined as

$$p = -\rho(\nu\Omega)P(\xi) + \frac{1}{2}\rho\omega_{core}^2 r^2 \quad (\text{B.5})$$

where ω_{core} is the angular velocity of the fluid in the core of the disk, which in general, for a rotor-stator cavity, is unknown. In the particular case of a rotating free disk $\omega_{core} = 0$. The static pressure is in the first approximation constant across the boundary-layer, and it exhibits a quadratic function with the radius, since $\omega_{core} \sim \Omega$.

Substituting Eqn. (B.3), Eqn. (B.4) and Eqn. (B.5) into the thin form of the axisymmetric incompressible Navier-Stokes Eqn. (B.1), we obtain:

$$\begin{aligned} 0 &= 0 + \frac{1}{Re_s^{1/2}} H' + 2F & [a] \\ \frac{\partial F}{\partial \tau} &= \frac{1}{Re_s} F'' - \frac{1}{Re_s^{1/2}} F' H + (G^2 - F^2) - \beta^2 & [b] \\ \frac{\partial G}{\partial \tau} &= \frac{1}{Re_s} G'' - \frac{1}{Re_s^{1/2}} G' H - 2FG & [c] \\ \frac{\partial H}{\partial \tau} &= P' + \frac{1}{Re_s^{1/2}} H'' - H'H & [d] \end{aligned} \quad (\text{B.6})$$

which represent, respectively, the dimensionless form of the continuity, radial momentum, circumferential momentum and axial momentum equations, where Re_s is the Reynolds number based of the disk spacing

$$Re_s = \frac{\Omega s^2}{\nu} \sim \left(\frac{s}{\delta_c} \right)^2 \quad (\text{B.7})$$

which physically represents the square of the ratio between the disk spacing and the boundary-layer thickness. If $Re_s^{1/2} \gg 1$ then the disk and stator boundary-layers are confined to a thin region of the cavity of the order of $\delta_c/s \sim Re_s^{-1/2}$. The swirl ratio, β , appearing in the radial momentum equation, is defined as:

$$\beta = \omega_{core}/\Omega \quad (B.8)$$

The origin of the swirl ratio β , appearing in the radial momentum equation, and which is actually unknown, is due to the injection of Eqn. (B.5) into the $\partial p/\partial r$ term of Eqn. (B.1)[b]. Because the fluid is considered as incompressible and with constant properties, the continuity and momentum equations decouple from the energy equation and the flow pattern can be discussed independently of the temperature field. In the work presented here buoyancy effects are disregarded, although they could be addressed following a similar formulation recurring to the Boussinesq approximation. This set of ODEs has to be integrated with the conditions, that at the rotating disk ($z = 0$) $\mathbf{v} = \Omega r \mathbf{e}_\theta$, and that at the stationary disk ($z = s$) $\mathbf{v} = 0$. We need to provide as well initial conditions for the problem compatible with the similarity hypothesis, hence at $t = 0$ we will assume that $\mathbf{v} = \omega_{core}^0 r \mathbf{e}_r$, where ω_{core}^0 is the angular velocity of the fluid at the initial instant. If we express these boundary and initial conditions in dimensionless form we obtain:

$$\begin{aligned} F = H = 0, \quad G = 1, \quad \text{at } \xi = 0 \\ F = H = 0, \quad G = 0, \quad \text{at } \xi = 1 \\ F \text{ or } H = 0, \quad G = \beta_0 \quad \text{at } \tau = 0 \end{aligned} \quad (B.9)$$

The problem reduces to one of finding the solution of Eqn. (B.6) [a] to Eqn. (B.6) [c], subject to the boundary conditions given in Eqn. (B.9). It is important to recall at this point that first, this system of equations needs just five boundary conditions, since the continuity equation does not contain any second derivative, but we have actually specified six. This extra boundary condition is used to determine the swirl factor β , which is actually unknown. Secondly, only two initial conditions are needed as the continuity equation does not contain any time derivative and the dimensionless radial and axial velocities are strongly coupled through the mass conservation equation.

B.1.1.2. Energy Equation

The energy equation of the axisymmetric form of the incompressible Navier-Stokes equations is:

$$\frac{\partial T}{\partial t} + v_r \frac{\partial T}{\partial r} + v_z \frac{\partial T}{\partial z} = \frac{k}{\rho c} \nabla^2 T + \frac{\mu}{\rho c} \Phi \quad (\text{B.10})$$

where c , is the specific heat, and Φ is Rayleigh dissipation function. If the thin form of the Navier-Stokes equations and Eqn. (B.2) are used, then $\nabla^2 T \simeq \partial^2 T / \partial z^2$. The temperature field has to satisfy the following boundary conditions:

$$T(\xi = 0, \tau) = T_R \text{ and } T(\xi = 1, \tau) = T_S. \quad (\text{B.11})$$

We will assume also that at $\tau = 0$, $T(\xi) = T_S$. The temperature can be decomposed as:

$$T(\xi, r, \tau) = T_0(\xi, \tau) + \left(\frac{r}{s}\right)^2 T_2(\xi, \tau) \quad (\text{B.12})$$

where the quadratic radial variation of the temperature is explicitly retained. The dimensionless temperature, $\tilde{T}$, can be written as

$$\tilde{T} = \frac{T - T_S}{T_R - T_S} = \frac{T_0 - T_S}{T_R - T_S} + \left(\frac{r}{s}\right)^2 \frac{T_2}{T_R - T_S} = \tilde{T}_0 + \tilde{r}^2 \tilde{T}_2. \quad (\text{B.13})$$

Injecting the previous expression in the energy equation and identifying terms in $\tilde{r}^0$ and $\tilde{r}^2$, we obtain, the following evolution equation for $\tilde{T}_0$:

$$\frac{\partial \tilde{T}_0}{\partial \tau} + \frac{1}{Re_s^{1/2}} H \tilde{T}_0' = \frac{1}{Pr Re_s} (\tilde{T}_0'' + 4\tilde{T}_2) + \frac{Ec}{Re_s} 12F^2 \quad (\text{B.14})$$

This has to be solved with the following boundary conditions, $\tilde{T}_0 = 1$ at $\xi = 0$ and $\tilde{T}_0 = 0$ at $\xi = 1$. The equivalent equation for $\tilde{T}_2$, reads:

$$\frac{\partial \tilde{T}_2}{\partial \tau} + \frac{1}{Re_s^{1/2}} H \tilde{T}_2' + 2F \tilde{T}_2 = \frac{1}{Pr Re_s} \tilde{T}_2'' + \frac{Ec}{Re_s} (G'^2 + F'^2) \quad (\text{B.15})$$

with the boundary conditions $\tilde{T}_2 = 0$ at $\xi = 0$ and $\tilde{T}_2 = 0$ at $\xi = 1$. The order of magnitude of the ratio of the viscous dissipation and the heat conduction is $Pr Ec$, where the Eckert parameter, is defined as:

$$Ec = \frac{\Omega^2 s^2}{c(T_R - T_S)} \quad (\text{B.16})$$

If the Eckert number is zero, or small, viscous dissipation is much smaller than the conduction and can be neglected. The solution of Eqn. (B.15) is in this case $\tilde{T}_2 = 0$, physically meaning that the heating due to the windage is negligible. It can be shown that $\tilde{T}_2 \sim Pr Ec \simeq Ec$, if $Pr \sim \mathcal{O}(1)$, as is actually the case for the air and other gases. In this case $\tilde{T}_2/\tilde{T}_0'' \sim Ec/Re_s \ll 1$, and then Eqn. (B.14) decouples from Eqn. (B.15), and can be solved independently. However the true relative weight of $\tilde{T}_0$ and $\tilde{T}_2$ depends on the dimensionless radius, $\tilde{r}$, which can actually be large, and hence, even if $\tilde{T}_2 \ll \tilde{T}_0$, $\tilde{T}_0 \sim \tilde{r}^2 \tilde{T}_2$. It is important to note as well that if $Ec/Re_s \ll 1$, viscous dissipation can be neglected as well in the equation for $\tilde{T}_0$, Eqn. (B.14). The energy equation is then linear and effects can be superimposed. The equation for $\tilde{T}_0$ can be decomposed into the solution of an homogeneous part, α , and a particular solution, K . The temperature field is then given by

$$\tilde{T}(\xi) = \alpha(\xi) + Ec K(\xi) + \tilde{r}^2 Ec \hat{T}_2(\xi) \quad (\text{B.17})$$

and the heat flux by

$$q = k \frac{(T_R - T_S)}{s} \left(\alpha_w' + Ec K' + \tilde{r}^2 Ec \hat{T}_{2w}' \right) \quad (\text{B.18})$$

with $\hat{T}_2 = \tilde{T}_2/Ec$. It is readily seen that only the first term of Eqn. (B.18) is proportional to ΔT , and that $\hat{T}_2$ is not controlled by any temperature difference, but by viscous heating.

B.1.1.3. Numerical Method

The set of non-linear time-dependent ODEs defined by Eqn. (B.6), Eqn. (B.14) and Eqn. (B.15) is discretized using a second-order method, both in space and time. Firstly, the radial momentum Eqn. (B.6) [b] is derived with respect ξ to eliminate its explicit dependence on the swirl factor, β

$$\frac{\partial F'}{\partial \tau} = \frac{1}{Re_s} F''' - \frac{1}{Re_s^{1/2}} F'' H - \frac{1}{Re_s^{1/2}} F' H' + 2GG' - 2FF' \quad (B.19)$$

and, secondly, the continuity Eqn. (B.6) [a], $F = -2Re_s^{1/2} H'$, is used to eliminate the dimensionless radial velocity, F , from the formulation. As a result, the following expression for the derivative of the radial momentum equation is obtained:

$$\frac{1}{Re_s} H^{(iv)} - \frac{\partial H''}{\partial \tau} = 4Re_s^{1/2} GG' + \frac{1}{Re_s^{1/2}} HH''' = R_H(H, G) \quad (B.20)$$

which has to be solved with the condition that $H = H' = 0$ at $\xi = 0$ and $\xi = 1$, The time derivative is discretized using second-order backwards differentiation:

$$\left(\frac{\partial H''}{\partial \tau} \right)^{n+1} = \frac{3(H'')^{n+1} - 4(H'')^n + (H'')^{n-1}}{2\Delta\tau} \quad (B.21)$$

A fully implicit method is constructed expressing the left hand side of Eqn. (B.20) at instant $n+1$ to obtain:

$$\frac{1}{Re_s} \left(H^{(iv)} \right)^{n+1} - \left(\frac{\partial H''}{\partial \tau} \right)^{n+1} = R_H^{n+1}. \quad (B.22)$$

Spatial derivatives are discretized using second-order central finite differences. The resulting system of non-linear algebraic equations is solved using a fixed point iteration method. The non-linear right hand side of Eqn. (B.22) is evaluated in the previous step of the fixed point iteration process, m , leading to:

$$\frac{1}{Re_s} \left(H^{(iv)} \right)_{m+1}^{n+1} - \left(\frac{\partial H''}{\partial \tau} \right)_{m+1}^{n+1} = R_H^{n+1}(G_m, H_m) \quad (B.23)$$

which requires the solution of a pentadiagonal system of linear equations, per iteration. The system is converged to machine accuracy before advancing to the next time step.

The circumferential momentum equation is solved analogously:

$$\frac{1}{Re_s} \left(G'' \right)_{m+1}^{n+1} - \left(\frac{\partial G}{\partial \tau} \right)_{m+1}^{n+1} = R_G^{n+1}(G_m, H_m) \quad (B.24)$$

Figure B.3.: Dimensionless radial (F) and circumferential (G) velocity distributions for a rotor-stator cavity rotating at $Re_s = 975$.

It has to be solved with the condition that $G = 1$ at $\xi = 0$, and $G = 0$ at $\xi = 1$. In this case Eqn. (B.24) represents a tridiagonal system of linear equations. Equations (B.23) and (B.24) are solved simultaneously. Once the velocity field is obtained, the energy Eqn. (B.14) and Eqn. (B.15) are solved using a similar approach, although in this case the fixed point iteration method is not needed since the energy equation is linear.

Numerical accuracy has been checked against well-known solutions for a free rotating disk with different swirl factors at infinity. Figure B.3 compares the dimensionless velocity distributions computed with the present method for two infinite disks, one rotating and the other stationary, at $Re_s = 975$, with the solutions obtained by Van Eeten et. al [147] and CFD results. It can be seen that the agreement between all of the cases is very good.

B.1.2. Results

The purpose of this section is to numerically and theoretically investigate the transient behaviour of a model laminar rotor-stator cavity at high Reynolds numbers using a similarity solution. Although it is well known that this type of flow rapidly experiences instabilities and transition to turbulence, still there is value in understanding different trends with the Reynolds number as a previous conceptual step to dealing with the more involved and realistic problem of turbulent

cavities. From a conceptual point of view, if Re_s is large enough, but smaller than the critical transition Reynolds number, the scaling laws for a laminar flow do not change. In the limit of high rotation rate, the flow between a stationary and a rotating disk presents two separated boundary-layers, the so-called Bödewadt and von Kármán boundary-layers respectively, which can transition to the turbulent regime at moderate local Reynolds number Re_ϕ^c . An important remark is that the effect of the local Reynolds number in the flow is independent of Re_s , as long as this is large enough to obtain separated boundary-layers ($Re_s \gtrsim 500$), see [131]. If $Re_\phi > Re_\phi^c$ the flow becomes turbulent and the scaling laws with Re_s change, but this point is not covered in the analytical work. For this reason we have decided to conduct most of the analyses at high, but modest, Reynolds numbers based on the gap, to capture the asymptotic trends with Re_s , but remaining within the laminar regime. The extrapolation to higher Re_s , but remaining in the laminar regime, is trivial.

B.1.2.1. Stationary solutions

Numerical results have been obtained for Reynolds gap numbers ranging between 600 and 30,000 to verify different trends with Re_s . The product of the Eckert number and the dimensionless radial distance based on the gap, $Ec \cdot \tilde{r}^2$, appearing in Eqn. (B.17) is the Eckert number based on the radius:

$$Ec_r = Ec \cdot \tilde{r}^2 = \frac{\Omega^2 r^2}{c(T_R - T_S)} \quad (\text{B.25})$$

Figure B.4 shows the numerical solution of the Batchelor flow for $Re_s = 2,000$, $Pr = 0.71$ and $Ec_r = 1$. For gas turbine applications, whose working fluid is air, the Prandtl number is fairly constant in a large temperature range, and therefore, its variation lacks interest and has not been pursued in this work.

Figure B.2 sketches the flow pattern within two infinite disks. Fluid is pumped outwards by the rotor motion through a thin layer, sometimes referred to in the literature as the von Kármán boundary-layer. This thin region is fed by a central part, where viscous effects are negligible, which is termed here the core region. The fluid in this core region, to a first approximation, rotates as a rigid body with constant angular velocity and null radial velocity. There is a second

Figure B.4.: Dimensionless velocity components (a) and temperature field components $\tilde{T}_0$ and $\tilde{T}_2$ (b) at $Re_s = 2,000$; $Pr = 0.71$; $Ec_r = 1$ and $\tilde{r} = 5$.

thin region in the stator, generally termed Bödewadt boundary-layer [148], with fluid moving inwards, to exactly compensate for the fluid expelled out by the rotor, finally enforcing the no slip boundary condition at the wall. The relative thickness of these boundary-layers with respect to the disk spacing is obtained balancing the convective and viscous terms in the momentum equation, $\delta_c/s \sim Re_s^{-1/2}$.

Figure B.4 (a) displays as well the temperature profile for the same case, where the three regions previously described for the velocity distribution can be noticed. The thermal boundary-layer thickness, δ_T , is of order $\delta_T/s \sim (Re_s Pr)^{-1/2}$, in our case, since $Pr \simeq 1$, both boundary-layer thicknesses are of the same order of magnitude, $\delta_T \sim \delta_c$. It can be seen that the dimensionless temperature and circumferential velocity profile are similar, which is a clear indication that the Reynolds analogy is fulfilled.

Figure B.4 (b) shows the temperature components $\tilde{T}_0$ and $\tilde{T}_2$ for the same case. According to the component $\tilde{T}_0$, the rotor can only heat a very thin region, of the order of $\delta/s \sim Re_s^{-1/2}$, close to the disk. It can be seen that $\tilde{T}_0 \simeq 0$ far away from the rotor. This is only possible because of the axial convection velocity, H , from the stator to the rotor, which prevents the diffusion of the rotor heat into the core region. The quadratic component of the temperature, $\tilde{T}_2$, is then the single mechanism to change the temperature of the core region by axial convection from the Bödewadt boundary-layer at the stator disk. As shown above for the previous assumptions,

a_θ	b_θ	c_θ	d_θ	e_θ	f_θ
0.5237	0.135	0.7501	0.2247	0.3593	0.2247

Table B.1.: Constant values for the non-dimensional conservation of angular momentum.

$\tilde{T}_2 \sim E_c$ and $\hat{T}_2 = \tilde{T}_2/E_c \sim 1$, so if there is no dissipation ($E_c = 0$), then $\tilde{T}_2 \simeq 0$ and $\tilde{T}_{core} \simeq 0$. It can be concluded that $\tilde{T}_{2,core}$ is created by a source in the immediate vicinity of the stator disk associated with the viscous dissipation, term $\frac{E_c}{Re_s}(G'^2 + F'^2)$ in Eqn. (B.15). Temperature in the core region is then a function of the Eckert number based on the radius, $\tilde{T}_{core} \sim E_{cr}$.

Angular Momentum Balance

The objective of this and the following section, is to link the differential solutions with an integral approach of the problem, which is frequently used in zero-dimensional models [149, 150, 151]. Moreover further insight can be gained by looking at the overall angular momentum and energy balance. The continuity equation, Eqn. (B.6)[a], and the momentum equation in the azimuthal direction, Eqn. (B.6) [c], can be combined to obtain the conservative form of Eqn. (B.6) [c],

$$\frac{1}{Re_s} \frac{\partial}{\partial \xi} (G') = 4FG + \frac{1}{Re_s^{1/2}} \frac{\partial}{\partial \xi} (HG) \quad (B.26)$$

Integrating Eqn. (B.26) over a control volume defined from the wall to a distance well beyond the edge of the boundary-layer, $\delta_K/s \gg Re_s^{-1/2}$ in the core region. The conservation of angular momentum in the overall domain can be separated into two independent control volumes, as shown in Fig. B.5. Making use of the fact that in the core region $F_{core} = G'_{core} = 0$, and at the wall $H_{wall} = 0$, the integral momentum equation becomes:

$$\underbrace{\frac{1}{Re_s} G' |_{rotor}}_{[1]} = - \underbrace{\int_0^{\delta_K} 4F(\xi)G(\xi)d\xi}_{[2]} - \underbrace{\frac{1}{Re_s^{1/2}} H_{core} G_{core}}_{[3]} \quad (B.27)$$

The dimensionless shear stress at the walls can be expressed as

Figure B.5.: Renormalised non-dimensional shear stress ($\tilde{\sigma}/Re_s^{1/2}$).

$$\begin{aligned}\tilde{\sigma}_{rotor} &= \frac{s}{\Omega r \mu} \sigma_{rotor} = \frac{\partial G}{\partial \xi} \Big|_{rotor} = a_{\theta} Re_s^{1/2} \\ \tilde{\sigma}_{stator} &= \frac{s}{\Omega r \mu} \sigma_{stator} = -\frac{\partial G}{\partial \xi} \Big|_{stator} = b_{\theta} Re_s^{1/2}\end{aligned}\tag{B.28}$$

where a_{θ} and b_{θ} represent the universal non-dimensional shear stress in the rotor and stator, respectively. In the same line $f_{\theta} = d_{\theta} = H_{core} G_{core}$, and c_{θ} is defined as

$$c_{\theta} = Re_s^{1/2} \int_0^{\delta_K} 4F(\xi)G(\xi)d\xi$$

in the rotor. The equivalent integral in the stator defines e_{θ} .

Figure B.5 outlines the tangential momentum equilibrium in a rotor-stator cavity identifying the terms of Eqn. (B.27), whereas Tab. B.1 provides the actual values of the constants used in this figure. The tangential momentum generated by rotation of the rotor, a_{θ} , is convected outwards, c_{θ} , together with the azimuthal momentum convected in the core from the stator to the rotor, d_{θ} . This actually means that $c_{\theta} = a_{\theta} + d_{\theta}$. Analogously, in the stator, momentum is radially convected inwards, e_{θ} , convected axially to the rotor, d_{θ} , and transmitted partially to the stator, b_{θ} . Momentum balances dictates that $e_{\theta} = b_{\theta} + d_{\theta}$. It is interesting to note that since $a_{\theta} \neq b_{\theta}$, the torque of the rotor is not fully transmitted to the stator.

Energy Balance

Analogously to the azimuthal momentum equation, the continuity Eqn. (B.6) [a], can be combined with the energy Eqn. (B.14) and Eqn. (B.15), to obtain the corresponding energy equa-

Figure B.6.: Renormalised non-dimensional heat flux balance $(qs/k(T_R - T_s)Re_s^{1/2})$.

tions in conservative form

$$\begin{aligned} \frac{1}{PrRe_s} \frac{\partial}{\partial \xi} (\tilde{T}'_0) &= 4F\tilde{T}_0 + \frac{1}{Re_s^{1/2}} \frac{\partial}{\partial \xi} (H\tilde{T}_0) & [a] \\ \frac{1}{PrRe_s} \frac{\partial}{\partial \xi} (\tilde{T}'_2) &= 4F\tilde{T}_2 + \frac{1}{Re_s^{1/2}} \frac{\partial}{\partial \xi} (H\tilde{T}_2) - \frac{Ec}{Re_s} (G'^2 + F'^2) & [b] \end{aligned} \quad (B.29)$$

Again, integrating Eqn. (B.29) over a control volume defined from the wall to a distance well beyond the edge of the thermal boundary-layer, $\delta_T/s \gg Pr^{-1/2}Re_s^{-1/2}$ in the core region. The conservation of energy in the whole domain can be split into two independent control volumes. Making use of the fact that, in the core region $F_{core} = T_{0,core} = T'_{0,core} = T'_{2,core} = 0$, and at the wall $H_{wall} = 0$, the two integral energy equations become:

$$\begin{aligned} \underbrace{\frac{1}{PrRe_s} \tilde{T}'_0|_{rotor}}_{[1]} &= - \underbrace{\int_0^{\delta_r} 4\tilde{T}_0(\xi)F(\xi)d\xi}_{[2]} \\ \underbrace{\frac{1}{PrRe_s} \tilde{T}'_2|_{rotor}}_{[1]} &= - \underbrace{\int_0^{\delta_r} 4\tilde{T}_2(\xi)F(\xi)d\xi}_{[2]} - \underbrace{\frac{1}{Re_s^{1/2}} H_{core} \tilde{T}_{2,core}}_{[3]} \\ &\quad + \underbrace{\frac{Ec}{Re_s} \int_0^{\delta_r} (G'^2 + F'^2)d\xi}_{[4]} \end{aligned} \quad (B.30)$$

Figure B.6 sketches Eqn. (B.30) in graphical form. Heat flux is non-dimensionalised using,

a_E	b_E	c_E	d_E	e_E	f_E	g_E	h_E	i_E
0.4283	0.294	0.1834	0.1878	0.1854	0.2964	0.3033	0.0711	0.1878

Table B.2.: Constant values for the non-dimensional energy budget for $Pr = 0.71$.

$q_{ref} = k(T_R - T_S)/s$, and renormalised with $Re_s^{1/2}$ to absorb the dependence with the Reynolds number. The non-dimensional heat fluxes at the rotor and stator are then:

$$\begin{aligned} Nu_{rotor} &= \frac{q_{rotor} \cdot s}{k(T_R - T_S)} = -\frac{\partial \tilde{T}}{\partial \xi} \Big|_{rotor} = (a_E - b_E Ec_r) Re_s^{1/2} \\ Nu_{stator} &= \frac{q_{stator} \cdot s}{k(T_R - T_S)} = \frac{\partial \tilde{T}}{\partial \xi} \Big|_{stator} = c_E Ec_r Re_s^{1/2} \end{aligned} \quad (B.31)$$

where the explicit dependence on $Re_s^{1/2}$ can be seen. This definition is consistent with the classical definition of the Nusselt number based on the gap. All the terms affected by the Ec_r have their origin in the quadratic terms of the energy equation. The rest of the terms of Tab. B.2 are defined as

$$\begin{aligned} d_E &= \frac{1}{Ec} H_{core} \tilde{T}_{2,core} \\ e_E &= \frac{Re^{1/2}}{Ec} \int_0^{\delta_T} 4\tilde{T}_2(\xi) F(\xi) d\xi \\ f_E &= \int_0^{\delta_r} (G'^2 + F'^2) d\xi \end{aligned} \quad (B.32)$$

The rest of the constants in Tab. B.2, c_E , i_E , h_E , and g_E , are the corresponding parts in the stator of b_E , d_E , f_E and e_E , respectively.

As can be shown in Fig. B.6, viscous dissipation has a determinant effect on the temperature near the rotor wall. In addition we can estimate the value of the temperature at the core region using the energy balance previously studied. Assuming the conservation of mass flow over the whole domain, the dimensionless axial velocity at the core region can be estimated as follows,

$$H_{core} = 1 - \beta^2 H_{Bödewadt} \quad (B.33)$$

keeping in mind that $H_{Bödewadt} = 1.35$ [148]. On the other hand, using the numerical results of the energy balance shown in Fig. B.6, we can derive d_E , and making use of the expression of the

axial convection term in Eqn. (B.32), we can estimate $\tilde{T}_{core}$ as a function of the dimensionless axial velocity at the core region as follows, using the constant values given in Tab. B.2,

$$\tilde{T}_{core} = \frac{d_E Ec_r}{H_{core}} \quad (B.34)$$

Substituting Eqn. (B.33) into Eqn. (B.34) we can obtain an estimate for $\tilde{T}_{core}$ as a function of the swirl factor and the Eckert number,

$$\tilde{T}_{core} = \frac{d_E Ec_r}{1 - 1.35 \beta^2} \quad (B.35)$$

Note that the radial variation of the temperature, $\hat{T}_2$, is not controlled by the temperature difference between the rotor and the stator, since actually $T_{2R} - T_{2S} = 0$. The temperature in the core, $\hat{T}_{core}$, is controlled by viscous heating in the stator and the subsequent convection from the stator to the rotor; windage heating contributes to increase the temperature at the core region. The increment of temperature due to the dissipation can be estimated as

$$\Delta T_{viscous} \sim \frac{\Omega^2 r^2}{c} \quad (B.36)$$

B.1.2.2. Transient solutions

Instantaneous Spin-Up

A former study by Owen et. al [152] demonstrated that the steady solutions obtained with the present formulation were able to reproduce, with reasonable agreement experimental results. In the present work emphasis is placed on the transient evolution of a still fluid when the rotor is impulsively started. The same set of dimensionless parameters used in the stationary case has been kept. Although the initial and final states of the fluid are relatively simple and have been well studied in the past, the transient state is non-trivial because it involves the time dependent evolution of a coupled convective and diffusive mechanical system.

Let us consider again the dimensionless circumferential momentum equation:

$$\begin{array}{ccccccc}
 \frac{\partial G}{\partial \tau} & + & 2FG & + & \frac{1}{Re_s^{1/2}} G'H & = & \frac{1}{Re_s} G'' \\
 [1] & & [2] & & [3] & & [4] \\
 \downarrow & & \downarrow & & \downarrow & & \downarrow \\
 \frac{1}{\tau_c} & & \frac{1}{Re_s^{1/2}} & & \frac{1}{Re_s^{1/2}} & & \frac{1}{Re_s}
 \end{array} \tag{B.37}$$

In the core region, $v_\theta \sim \Omega r$, and therefore $G \sim \mathcal{O}(1)$, the radial velocity is very small, $v_r \ll \Omega r$. Actually, in accordance with the dimensionless continuity equation, Eqn. (B.6)[a], $F \sim Re_s^{-1/2}$, and therefore both convective terms of Eqn. (B.37) (terms [2] and [3]) are of order $Re_s^{-1/2}$, and then much larger than the diffusive term [4], which is of order Re^{-1} . The time derivative is of the same order as the convective terms $[1] \sim [2]$ or $[3]$ when $\tau_c = \tau_r \sim Re_s^{1/2}$. Physically, this is the time needed for a fluid particle to move from the stator to the rotor, τ_r . This can be noticed by realising that the characteristic velocity in the axial direction is $v_{zc} \sim (v\Omega)^{1/2}$, and therefore

$$\tau_r = \Omega t_r = \Omega \left(\frac{s}{v_{zc}} \right) \sim \frac{\Omega s}{(v\Omega)^{1/2}} = Re_s^{1/2} \tag{B.38}$$

The dimensionless diffusive time, τ_d , corresponds to the time taken by the diffusion to transport angular momentum in the axial direction. This would be the characteristic time if H , i.e. the axial velocity, were strictly zero. In this case the balance between the temporal derivatives and the viscous terms (term [4] in Eqn. (B.37)) would yield $\tau_c = \tau_d \sim Re_s$. This conclusion can be directly obtained by non-dimesionalising the diffusion time

$$\tau_d = \Omega t_d \sim \Omega \left(\frac{s^2}{v} \right) = Re_s \tag{B.39}$$

There is a third characteristic time, which is the time taken by the diffusion to form the disk boundary-layer, $t_{BL} \sim \delta_c^2 / v = \Omega^{-1}$. This was the time taken to non-dimesionalise the equations, and hence, $\tau_{BL} \sim \mathcal{O}(1)$.

The existence of three different characteristic times gives rise to three distinct phases of mo-

Figure B.7.: Dimensionless velocity components at the midpoint of the core region ($\xi = 0.5$) as a function of dimensionless time scaled with the diffusive time at $Re_s = 2,000$ for a disk impulsively started from rest (middle). Velocity fields at; $\tau = 0.2Re_s$ (Stage II) and $\tau = 0.52Re_s$ (Stage III), left and right respectively.

tion, namely, the rapid formation of the von Kármán boundary-layer (Stage I) where a quasi-stationary Stewartson-type of flow is created; a spinning-up (Stage II) process by which the non-viscous core acquires its angular momentum, and finally the formation of a Batchelor-type flow field, once Bödewadt boundary-layer is created at the stator and the swirl at the core is adjusted, balancing the rotor and stator boundary-layers (Stage III).

Figure B.7 displays the time evolution of the axial and circumferential velocities for the impulsive start from rest ($F_0 = G_0 = H_0 = 0$ at $\tau = 0$) of one of the disks of an infinite rotor/stator cavity at $Re_s = 2,000$, computed with the numerical scheme previously outlined. At $\tau = 0$, the rotor is impulsively set in motion, resulting in the very rapid formation of a Stewartson-type flow. This phase (Stage I) can be seen as the development of a von Kármán boundary-layer in a free disk, which is independent of Re_s . The characteristic time of the Stage I is of order unity, $\tau_I \sim \mathcal{O}(1)$. A von Kármán boundary-layer is created in the rotating disk in which the azimuthal velocity decreases from the disk rotational velocity at the wall to the core velocity, which at this initial state is zero.

The subsequent Stage II, is highly sensitive to the details of the initial condition of the circumferential velocity. If G_0 is strictly zero at $\tau = 0$, then the azimuthal velocity can only increase by diffusion from the disk wall, since the convective terms, [2] and [3], in Eqn. (B.37) are strictly

Figure B.8.: Dimensionless time of Stage II and III for different initial conditions.

zero in the core region. The axial velocity, H , constant and negative in the core region, is responsible of feeding the radial boundary-layer created in the disk, however it can only transport fluid with zero tangential momentum. This effectively means that the tangential momentum created by the disk is convected back to the disk, as a consequence the tangential momentum in the core region can only be increased by diffusion, and hence this stage lasts for a time $\tau_{II} \sim Re_s$, which is very long. This can be seen in Fig. B.7 (left) where a Stewartson-like velocity profile is observed in this stage.

The quasi-steady Stewartson-type flow rapidly changes after $\tau \simeq 0.34Re_s$, see Fig. B.7 (middle). At this point the tangential momentum, transported by diffusion from the disk, is radially convected inwards, because the radial velocity in the core region of the Stewartson flow is negative. Since in the inviscid core region rv_θ is constant along the pathlines, a reduction in radius implies automatically an increase in the dimensionless circumferential velocity, G . From a mathematical point of view the term [2] of Eqn. (B.37) is not zero any more and triggers the exponential creation of circumferential momentum in a time $\tau_d = \tau_{II} \sim Re_s$. Some researchers [153] describe this phenomenon as an instability of the Stewartson flow.

Once the core of the disk begins to rotate as a rigid body, a Bödewadt boundary-layer is created in the stationary disk, in a time of the order of the diffusive time of the boundary-layer, $\tau_{BL} \sim \mathcal{O}(1)$. This can be seen in Fig. B.7 (right) where a Batchelor-type flow is depicted.

However during this Stage III, the von Kármán and the Bödewadt boundary-layers are not in equilibrium since they are not symmetric. For a given arbitrary swirl factor of the core, β , firstly, the tangential momentum created in the core by both boundary-layers is not in equilibrium, and secondly, the fluid ingested by the disk and egressed by the stationary disk do not match either. This global balance is established by convecting tangential momentum and mass from the stationary to the rotating disk in a time of the order of the residence time of the axial velocity of the core, $\tau_r = \tau_{III} \sim Re_s^{1/2}$. The axial mass flow ingested by the disk during the Stage II of the process is smaller than the egressed by the stator once Bödewadt boundary is formed and the final steady state is reached, see Fig. B.7 (middle), since the flow is incompressible, this mismatch gives rise to a steep change in the axial velocity at $\tau \simeq 0.52Re_s$, Fig. B.7 (right), and inertial waves are clearly seen in Fig. B.7 (middle). We have tracked the duration of the spikes seen during the third phase of the transient stage, and we have verified that its characteristic time scales also as $\tau_c \sim Re_s^{1/2}$.

We have solved the problem for different Re_s and identified manually Stages II and III. Figure B.8 displays the duration of the Stage II (◆) and Stage III (●) as a function of the Reynolds number. The slopes 1 and 0.5 are plotted to help the reader to draw their own conclusions. It is clearly seen that when the disk is impulsively started from rest, and the three components of the fluid velocity are strictly zero, the characteristic times of the Stage II and III of the problem scale with Re_s and $Re_s^{1/2}$, respectively. These trends are more easily seen at the largest Reynolds numbers.

Stage II of the transient changes dramatically if the dimensionless circumferential velocity is initialised to a small value. Figure B.8 displays the duration of Stage II when G is initialised to 10^{-4} (◇). It can be seen that, in this case, the Stage II is bypassed, and only the convective part of the process, which takes place in times of the order of $Re_s^{1/2}$, can be seen. The starting process described, when the fluid velocity is strictly zero at $\tau = 0$, cannot be seen in practice in real engines, since frequently analyses are focused on the acceleration of the engine from idle to maximum take-off. The flow in the initial stage is already rotating ($G_0 \neq 0$), and hence the diffusive stage is bypassed. Idle rotational speed is approximately 10% of maximum take-off

Figure B.9.: Transient evolution scaled with the diffusive time of the axial and circumferential dimensionless velocities (a) and dimensionless temperature (b) at the midpoint of the core region ($\xi = 0.5$) when the cavity is initially at rest ($G_0 = 0$ at $\tau = 0$) and when it is uniformly initialised ($G_0 = 10^{-4}$ at $\tau = 0$), at $Re_s = 4,000$; $Pr = 0.71$; $Ec_r = 1$.

rotational speed so that G_0 is actually much greater than 10^{-4} . Even if the disk is started from rest, there are always velocity perturbations associated either with the ventilation of the system or natural convection, if the engine is restarted, which are at least of the order of 10^{-3} . Hence our understanding is that the diffusive part of the process is always bypassed in realistic cases and it is not relevant.

It has been reported [154] that the spin-up and spin-down processes give rise to axisymmetric circular waves inside Bödewadt boundary-layer. These radial variations cannot be captured by the similarity analysis presented here, which prescribes a linear variation of the velocity with the radius. Fortunately, these perturbations die out in very short times, $\tau = \mathcal{O}(1)$. This actually means that this very short transient, although not seen in the solutions, hardly affects the global transient behaviour of the cavity, whose characteristic time is much longer.

Figures B.9 (a) and (b) compare the evolution, for the same Re_s , of the velocity and temperature fields in the core midpoint for two different initial conditions, $G_0 = 0$ and $G_0 = 10^{-4}$. It is clearly seen that the final state is reached much earlier when the flow is started up with a fairly small perturbation than when the fluid is initially at rest. It can be concluded that, for $G_0 = 10^{-4}$,

Figure B.10.: Transient evolution scaled with the diffusive time of the renormalised Nusselt number ($Nu/Re_s^{1/2}$) at the stator and rotor with $G_0 = 10^{-4}$, at $Re_s = 4,000$; $Pr = 0.71$; $Ec_r = 1$.

the global dimensionless time evolution can be estimated as

$$\tau_i \simeq \tau_I + \tau_{II} + \tau_{III} \propto Re_s^{1/2} \quad (\text{B.40})$$

Usually heat transfer analyses of the start-up process of a gas turbine assume that the flow is quasi-stationary with respect to the solid to avoid computing a very long transient stage, which can become unaffordable from a computational point of view. However we will show that there are analytical indications that demonstrate that this might not necessarily be the case.

The initial temperature of the fluid is supposed to be uniform and identical to that of the stator, $T_i(\xi) = T_S$. We study now the time-dependent evolution of the temperature field, when the disk is impulsively started and the fluid is at rest. As is shown in the time-dependent evolution of Nu_{rotor} in Fig. B.10, the thermal boundary-layer at the rotor is nearly instantaneously created (Stage I), in a dimensionless time of the order of unity, reaching a quasi-steady value of the heat flux at the rotor. Once the thermal boundary-layer at the rotor has been created, $\tilde{T}_0$ remains constant in the whole domain, and it is identical to that shown in Fig. B.4 (b) for the stationary case. The non-dimensional heat flux in the rotor, Nu_{rotor} , is constant and equal to that of the steady solution, for $\tau \gg 1$ (Stages II and III).

Figure B.11.: Dimensionless temperature field with $G_0 = 0$ (a) and $G_0 = 10^{-4}$ (b) at different dimensionless times for the case $Re_s = 4,000$; $Pr = 0.71$; $Ec_r = 1$.

The time-dependent evolution of the temperature field separating the contribution of $\tilde{T}_0$ and $\tilde{T}_2$, is studied in the following. The development of the temperature $\tilde{T}_0$ can be studied from Eqn. (B.14). Following the same argument as for the conservation of angular momentum in the azimuthal direction (Eqn. (B.6) [c]), it can be noticed that there is no term in Eqn. (B.14) which can create an exponential growth of the temperature field and the contribution of $\tilde{T}_0$ to the temperature field is restricted to a narrow region in the vicinity of the rotor. In Stage II, the temperature at the core region, which is solely due to the contribution of $\tilde{T}_2$, increases due to the heating associated with the viscous dissipation, term $\frac{Ec}{Re_s}(G'^2 + F'^2)$ in Eqn. (B.15). The linear profile of F gives rise to a constant source of heat. This can be seen in Fig. B.11 (a), where the instantaneous distribution of the temperature at $\tau \simeq 0.5Re_s$ exhibits a linear profile due to heating that a fluid particle undergoes while moving from the stator to the rotor. The transition of the temperature field from Stage II to Stage III is due to the appearance of radial convection of temperature, represented by the term $2F\tilde{T}_2$, in a dimensionless time of $\tau \sim Re_s$. This radial convection gives rise to the creation of Bödewadt boundary-layer with its associated dissipation and heat source. The enthalpy created in the stator region gives rise to a step change of temperature from $\tau \simeq 0.5Re_s$ to $\tau \simeq 0.6Re_s$, see Fig. B.11 (a). Finally the steady state is obtained approximately at $\tau \simeq 0.8Re_s$, where the thermal energy created in the stator is transported first to the rotor and then radially in the rotor boundary-layer.

If we consider now the case for which at $\tau = 0$, $G_0 = 10^{-4}$, analogously to what happened in

the velocity field, the diffusive (Stage II) is bypassed. As shown in Fig. B.11 (b) at $\tau \simeq 0.32Re_s$, the distribution of temperature along the axial direction is not constant, so when the Bödewadt boundary-layer is created, the global balance is established by axial and tangential convection, until it reaches the steady solution, as shown at $\tau \simeq 0.6Re_s$ in Fig. B.11 (b). In this case, simultaneously with the creation of the stator boundary-layer, the heat created in the stator region is transported to the disk, since the heating in the stator increases with time the slope of the temperature profile at $\tau = 0.32Re_s$ is negative.

Concerning the transient development of the thermal field, if we define the local Nusselt number, Nu_l , using the instantaneous bulk temperature of the core region, then

$$Nu_l = \frac{q_{wall} \cdot s}{k(T_R - T_{core})} = \frac{1}{1 - \tilde{T}_{core}(\tau, \tilde{r})} \cdot Nu \quad (B.41)$$

If we assume that Nu_{rotor} remains constant along time evolution (see Fig. B.10), since the temperature in the core varies with time (see Fig. B.9), the local Nusselt number previously defined increases with time.

Instantaneous Spin-Down

The transient evolution of the velocity and temperature in the core region of an infinite cavity between two disks, when initially one disk is rotating and the other is at rest, and the rotor is impulsively stopped, is now studied. The first stage of the process is the very rapid destruction of the von Kármán boundary-layer of the rotor and the creation of a new Bödewadt boundary-layer (Stage I) in dimensionless times of order unity, $\tau_I \sim \mathcal{O}(1)$. The azimuthal velocity profile increases from zero at the rotor wall (now at rest) to the core velocity, which is the one corresponding to the steady state before the spin-down.

An sketch of this initial stage is shown in Fig. B.12 (left). Once the circumferential velocity in the core of the cavity begins to decrease due to viscous effects (Stage II), the Bödewadt boundary-layer in the originally stationary disk is adapted to the new conditions outside of the boundary-layer. The axial velocity, positive in the left disk and negative in the right disk, is instantaneously changed from a negative value to a positive, which means that the convective

Figure B.12.: Dimensionless axial, circumferential velocities and temperature at the midpoint of the core region ($\xi = 0.5$) as a function of dimensionless time scaled with the diffusive time at $Re_s = 4,000$ for a speed-down regime where the rotating disk is suddenly stopped (middle). Velocity fields at; $\tau = 0.02Re_s$ (Stage II) and $\tau = 0.1Re_s$ (Stage III), left and right respectively.

axial component transports the temperature at the rotor to the core region until the solution reaches steady thermal equilibrium where $\tilde{T}_{core} = 0.5$.

During this stage, the two Bödewadt boundary-layers are not in equilibrium, since in general they are not symmetric due to the fact that the swirling factor of the core, β , is steadily decreasing with time. Both the tangential momentum and the axial mass flow are balanced by means of inertial waves which give rise to steep gradients in all the variables, especially in the axial velocity, as shown in Fig. B.12. The global balance of momentum and mass is continuously updated by axial convection in a time of the order of that needed by the fluid particles to move from the stator to the core, $\tau_r \sim Re_s^{1/2}$, giving rise to oscillations in the axial velocity.

The subsequent Stage III, is of a diffusive nature and its characteristic time is of the order of $\tau_t \sim Re_s$. The fluid reaches the actual rest state, although at the end of the Stage II it can be considered that the fluid velocity is zero. The most noticeable macroscopic effect during this phase is in the diffusion of the temperature field, until a linear temperature profile is attained in the cavity. However, although hardly seen in Fig. B.12, G and H also are smoothed out by diffusion. The characteristic time of this stage is $\tau_{III} \sim Re_s$, which is very long. This can be seen in Fig. B.12 (right).

B.1.3. Evaluation of Unsteady Effects

Impulsive start-up is usually considered an instantaneous process in which the disk reaches its final speed. In practice any ramp-up process occurs in a finite time. We consider here that the rotational speed of the rotor is increased linearly with time, departing from zero and reaching a stationary value. If we name the dimensionless speed-up time τ_s , we can write Ω as

$$\Omega = \frac{\tau}{\tau_s} \Omega_f \quad 0 \leq \tau \leq \tau_s \quad \text{and} \quad \Omega = \Omega_f \quad \text{if} \quad \tau > \tau_s \quad (\text{B.42})$$

The wall temperature, both at the rotor and the stator, is assumed to be constant. The change of the rotational velocity of the rotor gives rise to the variation of $Re_s(\tau)$ and $Ec_r(\tau)$ with time, reaching their final value, $Re_{s,f}$ and $Ec_{r,f}$, at the dimensionless time τ_s .

The objective of this subsection, and of the previous analyses, is to elucidate when the heat transfer at the wall can be computed using a quasi-steady approximation during a disk transient. Fully coupled fluid-solid transient analyses of realistic gas turbine cavities have been computed in the past [155, 156].

To reduce the computational cost the transient behaviour of the solid is computed, but frequently the fluid is considered quasi-stationary. In other words, a series of steady simulations with varying boundary conditions is computed, where the time plays the role of a parameter since time derivatives are neglected in the fluid. The objective of this work is to determine if this quasi-steady hypothesis holds.

Figure B.13 displays the transient evolution of our model rotor-stator cavity for $Re_{s,f} = 4,000$. The conclusions derived from this relatively low Re_s can be readily extended to higher, and more realistic, Re_s , provided that the flow remains laminar. The non-dimensional temperature in the core is represented as a function of the non-dimensional time, $\tau/Re_s^{1/2}$. The reason for this is that the non-dimensional characteristic time of the evolution of an initialised cavity is, $\tau_c \sim Re_s^{1/2}$.

Ramps of different duration have been considered, $\tau_s/Re_s^{1/2} = 12, 36$ and 60 . Lines represent the true transient evolution of the problem, whereas solid symbols correspond to steady solu-

Figure B.13.: 1D-Analytical Model: Time evolution scaled with the convective time of dimensionless temperature at the core region ($\xi = 0.5$) for different dimensionless speed-up times. ($Re_{s,f} = 4,000$; $Pr = 0.71$; $Ec_{r,f} = 1$).

tions for the instantaneous values of $Re_s(\tau)$ and $Ec_r(\tau)$. If the time derivatives were negligible, the solid symbols should be superimposed on the different lines. The vertical lines in the same figure mark the end of the ramp. It can be seen that for very short times, for which $Re_s(\tau)$ and $Ec_r(\tau)$ are relatively low, the evolution of the $\tilde{T}_{core}$ exhibits a weird behaviour. We do not intend here to describe this complex process involving very thick boundary-layers to avoid missing the whole picture of the problem. For the fastest ramps ($\tau_s/Re_s^{1/2} = 12$ and 36) the quasi-steady solutions have little to do with the true transient solution. For $\tau_s/Re_s^{1/2} = 12$ the temperature in the core is largely over-predicted, whereas for $\tau_s/Re_s^{1/2} = 36$, the last part of the ramp follows the quasi-steady hypothesis. The slowest ramp ($\tau_s/Re_s^{1/2} = 60$) matches, however, reasonably well with the quasi-steady solution. It is concluded, as expected, that unless $\tau_s \gg Re_s^{1/2}$, the quasi-steady hypothesis is not valid. In physical terms this means that

$$t_{ramp} \gg t_c = \frac{s}{(\Omega v)^{1/2}} = \Omega^{-1} Re_s^{1/2} \quad (\text{B.43})$$

where t_{ramp} is the acceleration time of the ramp. Typically the acceleration time of a large gas turbine is of the order of a few seconds. If we assume that $Re_s \sim 10^6$ and $\Omega \sim 2,000$ rpm, then $t_c \simeq 5s$. It is therefore concluded that the use of a quasi-steady approach is questionable for this

kind of application.

B.2. Comparison with two-dimensional axisymmetric simulations

In order to validate the simplified numerical model, several two-dimensional unsteady simulations at different Reynolds numbers were performed. The flow solver, known as Mu^2s^2T [107, 108], is based on a second order discretization, both in space and time, of the Navier-Stokes equations. Turbulence models were switched off to force the flow remain laminar. A confined finite rotor-stator cavity with a non-rotating casing with a wall temperature T_S , located at the rim of the cavity, is considered. The cavity includes the axis and the outer radius of the cavity is $r_f = b$. A 450×100 mesh in the (r, z) frame proved to be sufficient for the calculation of the rotor/stator cavity with a gap to radius ratio of $Y = s/b = 0.2$. The Reynolds number based on the radius, $Re_\phi = Y^{-2}Re_s$, is large ($Re_\phi \simeq 10^5$) but the flow is assumed to remain laminar (Regime II according to Daily et. al [127]), and hence the stator and rotor boundary-layers are separated by a core inviscid region.

The presence of a cylindrical casing at a finite radial distance has an important effect on the transient and steady behaviour of the cavity. It can also alter the global stability of the flow, causing major disruptions of the layer, possibly unrelated to the stability of the Bödewadt boundary-layer [157]. Lopez et. al [129] have reported that a confined rotor/stator cavity with $Y = 0.2$ and a rotating shroud is linearly stable with respect to axisymmetric perturbations, at least up to $Re_\phi \simeq 10^5$. The results presented here are obtained using an axisymmetric formulation of the problem so that the appearance of three-dimensional spiral waves is avoided by construction.

B.2.1. Steady solution

The influence of a stationary outer shroud on the steady state of the core region of a square cavity at high Reynolds numbers is displayed in Fig. B.14. In the inner region of the cavity, and up to $x \simeq 0.5$, both the dimensionless circumferential velocity, see Fig. B.14 (a), and temperature,

Figure B.14.: Influence of a stationary outer shroud on the steady laminar flow ($Re_s = 4,000$, $Re_\phi \simeq 10^5$) at the midpoint of the core region ($\xi = 0.5$) of a square cavity with a gap ratio $Y = 0.2$. (a) Dimensionless azimuthal velocity. (b) Dimensionless temperature. ($Pr = 0.71$, $Ec = 8 \times 10^{-2}$)

see Fig. B.14 (b), agree very well with the self-similar solution, where the dimensionless radial distance is defined as $x = r/b$. This fact had been previously reported for the velocity field [135, 136], but here it is shown that the same is true as well for the thermal field. The circumferential velocity in the inner region varies linearly with the radius, Eqn. (B.44), and the swirl factor at the core is $\beta \simeq 0.31$ [137], which corresponds to the value expected for a high Reynolds laminar flow, as it has been previously mentioned.

$$\frac{v_{\theta,core}}{\Omega b} = \frac{\omega_{core} r}{\Omega b} = \beta x \quad (\text{B.44})$$

Concerning the temperature distribution along the radius in the core, see Fig. B.14 (b), the self-similar and CFD solutions match very well up to $x \simeq 0.6$, but for higher values of the radius the effect of the outer stationary casing spoils the self-similar solution, as could be expected. Taking into account that $\tilde{r} = x/Y$, we can express the temperature in the core as a function of the Ec number, Eqn. (B.35), as

$$\tilde{T}_{core} = \frac{d_E Ec}{1 - 1.35\beta^2} \left(\frac{x}{Y}\right)^2 \quad (\text{B.45})$$

Figure B.15.: Dimensionless azimuthal velocity, $v_\theta/\Omega b$, and temperature, $\tilde{T}$, contour plots on a finite laminar rotor-stator cavity at two different dimensionless time; (a) and (c) $\tau = 4$; (b) and (d) $\tau = 22$. ($Re_s = 4,000$; $Pr = 0.71$; $Ec = 8 \times 10^{-2}$; $Y = 0.2$)

which corresponds to the quadratic trend of temperature that is seen in Fig. B.14 (b). If $Ec_b \sim \mathcal{O}(1)$, as is the case here ($Ec_b \simeq 2$), then $\tilde{T}_{core} \sim \mathcal{O}(1)$. For small values of the radius $Ec_r \ll 1$, the fluid cannot be heated by viscous dissipation, as is also shown in Fig. B.14 (b).

B.2.2. Unsteady solutions

This subsection aims at assessing the impact of a stationary outer shroud on the transient behaviour of an enclosed cavity using axisymmetric laminar analyses. As discussed by Van Eeten et al [153], the presence of the outer casing has a relevant impact on both the steady and the transient solutions of the problem. This casing imposes a zero or 1 swirl factor, depending on whether it is rotating or not, and spoils the self-similar solution in the vicinity of the outer casing, which has a constant swirl factor along the radius. The transient start-up process of the cavity has three distinct phases, the second of them differing from that identified in the self-similar one-dimensional analysis. Once the rotor is impulsively set into motion, in a very short time, of the order of the diffusive time ($\tau_l \sim \mathcal{O}(1)$), the von Kármán boundary-layer is created in the rotor, and the flow inside of a very thin boundary moves radially outwards, until it impinges in the

outer casing. The flow is then deflected horizontally, transporting the tangential momentum and enthalpy which was previously created in the disk boundary-layer to the core, see Figs. B.15 (a) and (c). The inwards flow, of the already existing Stewartson-type flow, transports the angular momentum and enthalpy located in the outer core region of the cavity. Since the characteristic radial velocity in the core is $v_{rc} \sim \Omega r Re_s^{-1/2}$, the characteristic time needed for a particle to move from the outer radius of the disk, b , to a radius r is $\tau_{II} \sim Re_s^{1/2} \ln(b/r)$, which is large since the radial velocity is very small. This phase of the transient development does not exist in the one-dimensional model, since in this approach the swirl factor along the radius is deemed to be constant during the transient period and the velocity field is assumed to vary linearly with the radius. This phenomenon triggers the convective transport term of angular momentum, as shown in Figs. B.15 (b) and (d). During this phase, and in the upper region of the cavity, the inwards flow is in the core, not in the stator boundary-layer, which at this stage has not been created yet. Finally, when at a certain radius the circumferential velocity reaches a finite value, Batchelor flow is created, and there is a third phase in which the fluid is transported from the stator to the rotor. This time is also long and this phase, in the core of the cavity, is similar to that described in the one-dimensional model ($\tau_{III} \sim Re_s^{1/2}$), and is associated with the convection of fluid from the stator to the rotor. It is concluded that, in a laminar enclosed cavity the characteristic time of the transient is $\tau \sim Re_s^{1/2}$, and the long diffusive phase identified in the one-dimensional model is always bypassed.

The practical relevance of the transient analysis of the flow in a rotor-stator cavity has mainly to do with its impact in the heat exchange between the cavity and the disks. Let us consider the energy transfer evolution during the impulsive start of one of the two coaxial disks enclosing a fluid cavity. The energy equation of the disk in integral form is:

$$\bar{\rho}V \frac{dE}{dt} = \underbrace{\int_r k \nabla T \cdot dA}_{Q_{rotor}} + \underbrace{\int_s k \nabla T \cdot dA}_{Q_{stator}} + \underbrace{\int_c k \nabla T \cdot dA}_{Q_{casing}} + \underbrace{\int_r (\tau' \cdot v) \cdot dA}_{Q_{windage}} \quad (B.46)$$

where $\bar{\rho}$ is the mean density of the cavity, E is the total energy, V its volume, Q_{rotor} , Q_{stator} , Q_{casing} the energy transferred by conduction between the fluid and the rotor, stator and cas-

Figure B.16.: Transient evolution scaled with the convective time of the components of the energy balance of a rotor-stator cavity with gap ratio $Y = 0.2$, represented by Nusselt number based on the outer radius. Case of $Re_s = 4,000$; $Pr = 0.71$; $Ec = 8 \times 10^{-2}$.

ing respectively and $Q_{windage}$ is the energy transferred from the disk to the fluid by viscous friction. In order to numerically show the characteristic time to reach the steady state, we non-dimensionalise the energy in the follow way

$$\hat{Q} = \frac{Q \cdot s}{k(T_R - T_S)\pi b^2} \quad (B.47)$$

Figure B.16 displays time evolution of the different terms of the energy Eqn. (B.46) of the cavity in integral form. The dimensionless time needed to reach the steady state in the cavity is approximately $\tau_t \simeq 5.2Re_s^{1/2}$. The heat flux at the rotor and the mechanical energy transferred from it to the fluid, are nearly constant after a very short time. The same is true for the outer casing which quickly reaches its final state once the hot fluid coming from the rotor boundary wets the shroud. The heat flux at the stator departs from zero and increases when the core of the cavity is heated with hot fluid coming from the outer shroud due to the inwards radial convection at the core. This process takes place with the simultaneous creation of the Bödewadt boundary-layer in the stator. It is concluded, as can be seen in Fig. B.16, that the characteristic time of the transient of this two-dimensional model cavity is $\tau_c \propto Re_s^{1/2}$, exactly as in the one-dimensional model, although for different physical reasons.

Figure B.17.: Comparison of the non-dimensional circumferential velocity profiles obtained from 2D numerical simulations (Solid symbols) and from the simplified similarity analysis (Filled symbols) at different time instants at the same dimensionless radius ($x = 0.45$).

B.2.2.1. CFD comparison with one-dimensional Analytical Model

It can finally be questioned whether the flow obeys the similarity analysis during the spin-up or spin-down transients. The similarity solution imposes a strong constraint on the radial variation of the velocity ($v_r \sim \Omega r$ and $v_\theta \sim \Omega r$) which cannot capture the radial waves identified by Lopez et. al [154] in the Bödewadt boundary-layer during the spin-down of a square cavity. However since these perturbations die out in very short time ($\tau \sim \mathcal{O}(1)$), we assume that this process hardly affects the overall transient of the cavity whose characteristic time is much longer ($\tau \sim Re_s^{1/2}$).

In order to assess the similarity hypothesis, Fig. B.17 compares the dimensionless circumferential velocity profiles obtained by the one-dimensional (1-D) similarity analysis and 2-D numerical simulations during the instantaneous spin-up of a square cavity at different time instants for the same radius ($x = r/b = 0.45$). The time instants of the 1-D and 2-D solutions are different because the physics of the transient phase of the shrouded cavity is different. Moreover we have constructed the figure with the profiles obtained from the similarity analysis when the fluid is initially at rest ($G_0 = 0$). In order to ease the comparison with the results obtained in the previous section for the 1-D self-similar analysis, we express the dimensionless time scaled with Re_s , although we stress that the time development of Stage II in an enclosed cavity is dominated

Figure B.18.: Time evolution scaled with the convective time of the dimensionless temperature at the core region ($\xi = 0.5$) of a two-dimensional rotor-stator closed cavity of aspect ratio $Y = 0.2$ at a dimensionless radius $x = 0.4$ for different dimensionless speed-up times. ($Re_{s,f} = 4,000$; $Pr = 0.71$; $Ec_{r,f} = (\frac{x}{Y})^2 \cdot Ec_f = 0.32$).

by convective transport of angular momentum from larger to shorter radii, and that the characteristic time of this process is $\tau \sim Re_s^{1/2}$. It can be noticed that at $x = 0.45$, the velocity profiles of the 2-D analysis resemble those obtained in the 1-D analysis, and that the transient can be seen as a sequence of Stewartson ($\tau = 0.01Re_s$) and Batchelor ($\tau \geq 0.032Re_s$) type solutions. The core region is present from the very early stages of the process and the swirl factor in the core steadily increases from zero to its final stationary value. The actual times of the snapshot are however quite different, as has been already mentioned, because the start-up processes have a completely different nature.

Figure B.18 displays the speed-up process of the same enclosed rotor-stator cavity from rest to a final Reynolds number based on the gap $Re_{s,f} = 4,000$, for different acceleration times of the disk. The flow remains laminar along the whole process since the final Reynolds number based on the radius is $Re_{\phi,f} \simeq 10^5$. The counterpart for the 1-D model is Fig. B.13. The qualitative behaviour for the slowest ramps of the 1-D Model (Fig. B.13) and the 2-D model (Fig. B.18), match reasonably well with the quasi-steady solution. The slowest non-dimensional accelera-

tion times are $\tau_s/Re_s^{1/2} = 60$ for the 1-D model, and 20 for the CFD model, reflecting the fact that the response of the square cavity is faster than that of ideal infinite cavity. As previously mentioned, the non-dimensional characteristic time of evolution is of order $\tau_c \sim Re_s^{1/2}$ for both cases and unless $\tau_s \gg Re_s^{1/2}$, the quasi-steady hypothesis is not valid for a realistic enclosed cavity or for an infinite cavity. This condition, in general, is not fulfilled for transients in gas turbine representative cases.

B.3. Summary

The flow field and heat transfer behaviour of a model rotor-stator cavity using the von Kármán self-similar analysis has been revisited in this appendix. The resulting system of non-linear time-dependent ODEs has been solved numerically using a second-order method, both in space and time. Numerical results have been obtained for Reynolds numbers ranging between 600 and 30,000 to derive different trends for high Reynolds numbers.

Fundamental concepts and practical conclusions of the flow patterns and heat transfer mechanisms have been drawn. The fluid is considered as incompressible with constant properties, and therefore the flow pattern can be discussed independently of the temperature field, if buoyancy effects are neglected. It is concluded that the disk can only heat a very thin region, of the order of $\delta/s \sim Re_s^{-1/2}$, close to the rotor. This is due to the fact that the axial convection from the stator to the rotor shields the core from the rotor and prevents its heating. The component of the temperature which changes quadratically with the radius, $\tilde{T}_2$, which is of order $\tilde{T}_2 \sim PrEc$, is then the single mechanism changing the temperature of the core region by axial convection from the Bödewadt boundary-layer to the rotating disk. It can be concluded that the temperature increases at the core region ($\tilde{T}_{core} \sim Ec_r$) is created by a heat source in the immediate vicinity of the stator disk, due to viscous dissipation. The integral angular momentum and energy balance of the cavity have been investigated to fully understand the balance between the stator and the rotor and to estimate the stationary solution. Non dimensional heat flux at the rotor is only a function of the Reynolds number based on the gap, and the Eckert number based on the radius

Ec_r , which accounts for the effect of viscous dissipation. The working fluid has been assumed to be air, and hence the dependence with the Prandtl number has not been pursued in this work.

Finally the time-dependent solution predicted by the self-similar analysis has been compared against 2-D axisymmetric laminar simulations, accounting for the effect of outer shroud of the cavity. It is concluded that, the one- and two-dimensional models describe qualitatively the same physics up to 50% of the radius of the cavity, whereas the outer part of the cavity is heavily influenced by the presence of the casing. The relevance of the transient analysis of the flow in an infinite rotor-stator cavity and its associated characteristic times have been assessed. Viscous and thermal diffusion gives rise in an infinite cavity to very long transients ($\tau_{II} \sim Re_s$) when the cavity is initially at rest, then a third and shorter phase ($\tau_{III} \sim Re_s^{1/2}$) dominated by the axial convection takes place. The long diffusion phase is bypassed when the flow is initialised with a small perturbation. The same is true in real enclosed cavities, where the presence of an outer shroud at a finite radial distance changes the dynamics of the process, In this case there is an inwards radial convection departing from the outer shroud. It has been shown that the characteristic time of this phase is also $\tau_c \sim Re_s^{1/2}$, and it is associated to the radial convection. It is concluded that in realistic applications the diffusive phase of the process can be rarely seen.

For large Reynolds numbers, only when the acceleration time of the disk is much greater than $s/(\Omega v)^{1/2}$ does the hypothesis that the flow is quasi-stationary holds. This hypothesis is quite often not fulfilled. It is concluded that the flow unsteadiness cannot be disregarded in the simulations, and that the validity of quasi-steady methodologies used in gas turbine representative problems is either doubtful or not appropriate.

The flow in finite rotor-stator cavities very rapidly develops instabilities leading to turbulence. It should be recalled that the conclusions derived in this work are valid as long as the flow remains stable and laminar. The characteristic times for a turbulent flow are different. However, here, the purpose was not to undertake an exhaustive investigation of all of the effects, but to create a building block of knowledge aimed at increasing the understanding of cavity flows in more complex situations under the assumptions made on this manuscript. The effect of transition and turbulent mixing have to be certainly taken into account in real applications, and it is speculated

that the quasi-steady assumption may fail also in this case, but under different conditions.