UNIVERSIDAD POLITÉCNICA DE MADRID

ESCUELA TÉCNICA SUPERIOR DE
INGENIEROS AERONÁUTICOS

Grid Generation and Fluid-Solid Coupling
Methods for the Investigation of Gas Turbine
Rotor Blade Internal Cooling

TESIS DOCTORAL

Zhi Wang

Máster en Ingeniería Aeronáutica

Madrid, 2017

Departamento de Motopropulsión y
Termofluidodinámica

Escuela Técnica Superior de Ingenieros Aeronáuticos

Grid Generation and Fluid-Solid Coupling
Methods for the Investigation of Gas Turbine
Rotor Blade Internal Cooling

Autor

Zhi Wang

Máster en Ingeniería Aeronáutica

Director

Roque Corral García

Doctor Ingeniero Aeronáutico

Madrid, 2017

Escuela Técnica Superior de Ingenieros Aeronáuticos

UNIVERSIDAD POLITÉCNICA DE MADRID

Tribunal nombrado por el Magnífico y Excelentísimo Señor Rector de la Universidad Politécnica de Madrid. El día de de 2017.

Presidente: D. Benigno Lázaro Gómez

Secretario: D. Francisco Higuera Antón

Vocal: D. Li He

Vocal: D. José Ignacio Nogueira Goriba

Vocal: D. Ignacio Mayo Yague

Suplente: D. Raquel Gomez Miguel

Suplente: D. Manuel Antonio Burgos Olmos

Realizado el acto de defensa y lectura de la tesis el día de de 2017 en la E.T.S.I. Aeronáuticos.

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 Industria de Turbo Propulsores, S. A.. He has been providing many guidance and advices in all time of my study, despite he was extremely busy as a chief and senior fellow of the Technology & Methods Department of ITP. His immense knowledge and enthusiasm in engineering work and research are worth to learn for my entire life.

My thanks are dedicated to the directors and work colleagues in the Advanced Engineering Direction of ITP, for their supports to work as a part of this talent team. I would specially thank Juan Mauricio Chia and Jaime Quintanal for sharing their great experiences in programming and co-working in grid generation systems; Jose Maria Chaquet and Guillermo Pastor for their helps in heat transfer methods, Jesus Pueblas and Fernando Gisbert for giving me many advices in performing numerical simulations; Jose Javier Alvarez for his knowledge in the internal cooling system design of turbine blade, Eduardo Garcia for the management in cluster systems. And many more, Juan Manuel Gallardo, Oscar Cordoba, Olga Garcia, Ana Puig, etc. I received so many helps from all of you, in work, and in life. These years of working experience in ITP will certainly be one of the most valuable and unforgettable time.

My sincerely thanks go to other members outside of ITP. Prof. Carlos Martel from UPM, who

has been helping me solving enormous paperwork in the university. Dr. Francois Chedevergne from ONERA, for his co-working and advise in the European ERICKA project. I would like also to thank my best Chinese friends in Madrid, Dr. Ganggao Zhu, Bo Zhu, Yuntian Ji, Dr. Shan Huang, Dr. Jiayao Liu, Dr. Jie Song, Dr. Yu Hu, Dr. Meijuan Zhang, etc. There would not be wonderful life in Spain if I had never not meet so many friends here.

I would like to thank my family. Words can not express how grateful to them. I am very thankful to my parent and Dr. Yueru Zhong for their unconditionally support and encouragement in my life, especially when we were half-earth apart. Their loves and understanding sustained me thus far.

Last but not the least, I would like to thank all the jury members: Prof. Benigno Lázaro, Prof. Francisco Higuera, Prof. Li He, Prof. José Ignacio Nogueira and Dr. Ignacio Mayo for their presence, evaluation and providing many valuable advices to my work.

The research leading to these results has received funding from Industria de Turbopropulsores, S. A. and the European Union Seventh Framework Programme ERICKA (grant agreement No. 233799). The author wishes to thank ITP for its support and allowing me to publish the results.

Abstract

With the rapid growth of computer power, numerical methods are available and showing their advantages in identifying the complex underlying physics of flows. However, they have encountered a number of challenges in internal cooling problems of gas turbine rotor blade due to the complexity of some characteristics of rotating passages, such as geometries, flow conditions and heat transfer behaviours under rotations. This thesis focuses on the challenges in the numerical grid generation, coupling fluid/solid heat transfer method and understanding the rotational effect on flow and heat transfers.

Mesh generation is a critical process in Computational Fluid Dynamics (CFD) since mesh quality can not only have a direct impact on the final solution, but become a bottleneck of the whole analysis process. Grid spacing should be smoothly and sufficiently refined in certain regions to properly resolve steep solution gradients, whereas it should be coarser in other regions not compromised by mesh resolution considerations. A key element to achieve good quality meshes in viscous flow simulations at high Reynolds numbers is a high enough mesh resolution near solid walls to properly capture boundary layer gradients in the wall-normal direction. These regions account only for a small fraction of the computational domain but are directly linked with the quality of aerodynamic simulations. However, significant difficulties arise in the presence of complex geometric features such as multiple concave/convex corners, micro-surfaces, etc., which may result in mesh generation failure, or considerable elapsed time of the mesh generation process with the subsequent associated labour cost. Therefore, in an industrial environment, the generation of viscous mesh still remains a challenging task. The present work proposed a robust and efficient viscous mesh generation method to construct the computational grid in a

hybrid manner in order to mesh the complex 3D configurations efficiently for CFD analysis. The method first focuses on providing a robust criteria for the validity checking procedure for complex problems, and then improves the efficiency by implementing a pre-processing scheme to reduce the number of nodes that require to be checked. Therefore, the methods substantially reduces the generation time. The presented algorithm is proved to be much more efficient than common commercial or open source codes with equal or higher robustness.

The accuracy of temperature prediction is of paramount importance for turbine analysis, since an accurate prediction can prevent local hot spots and increase turbine blade life, avoiding at the same time the use of excessive cooling air and therefore allowing higher turbine inlet temperature. Traditionally, CFD and solid thermal conduction analyses have been performed separately at different stages of the design loop. Several hypotheses are implicit in this process and the accuracy of this kind of methods inevitably relies on empirical database and engineering judgement. Therefore, in highly complex cases, traditional methods may lead to considerable errors in the estimate of the temperature field, and to severe logistic problems to handle the exchange of information among different codes. Greater accuracy is offered by coupled fluid-solid heat transfer methods, in which fluid and solid heat transfer problems are solved simultaneously without the use of further hypotheses, since the heat flux and temperature continuity across fluid-metal interface is ensured by construction. Because of their relevance, coupled heat transfer algorithms have received considerable attention from different engineering perspectives in the last decades. However, the main disadvantage of coupled heat transfer methods is the relatively large computing times compared to that of an uncoupled stand-alone CFD simulation. Therefore, it is difficult to apply in practical engineering applications. This thesis proposes a fast and robust steady state loosely coupled fluid-solid heat transfer method based on the improvement of a loosely Dirichlet-Robin coupling approach to reduce the computational time cost. This method has been validated and applied with representative turbomachinery cases. The results have shown that the computational time has been substantially reduced compared to the baseline method, and it only spends around 35% more than a steady state CFD simulation, which is attractive for industrial applications

Rotating flows can be met in aero-engines due to system rotation. The rotating flows under Coriolis and centrifugal forces affect the flow patterns drastically compared to stationary cases. It is difficult to carry out real design without a deep understanding of the physics of rotating flows. The last part of this work provides the understanding of the rotational effect in the internal cooling channels of turbine rotor blade. The flow and heat transfer behaviour of an engine representative rotating channel has also been investigated and presented as an example of the numerical method in real applications.

Keywords: viscous grid generation, hybrid mesh, coupled heat transfer method, rotational effect, internal cooling channel, heat transfer

Resumen

Gracias al importante crecimiento de la potencia de cálculo los métodos numéricos se han convertido en una herramienta muy ventajosa a la hora de indentificar la complejidad física que subyace en los flujos. En el estudio de los problemas de refrigeración interna de los álabes del rotor en las turbinas de gas aparecen, todavía, desafíos importantes debido a las características que reúnen este tipo de elementos: complejidad de las geometrías, de las condiciones de flujo y de la naturaleza de la transferencia de calor en pasajes rotatorios. Esta tesis centra su estudio en en los desafíos presentes en el campo de la generación de malas numéricas, el método de transferencia de calor acoplando modelos fluido/sólido y en la comprensión de los efectos de la rotación en el flujo y en la transferencia de calor.

La generación de mallas es un proceso crítico en el campo de la Dinámica de Fluidos Computacional no sólo porque la calidad de las mismas afectará a la solución final, sino porque su propia generación supone un reto que afecta a todo el ciclo de análisis. La discretización debe permitir capturar de manera adecuada los complejos gradientes que se producen en determinadas zonas, bien por la propia geometría de las mismas o bien por la naturaleza física del flujo ahí presente. Por tanto, la malla será suficientemente fina y suave en estas zonas mientras que será más basta en aquellas otras zonas en las que una discretización excesiva no aporta una mejor solución. En simulaciones de flujos viscosos para altos números de Reynolds, disponer de una malla de buena calidad que permita capturar los gradientes de capa límite en dirección normal a la pared es un punto clave. Para ello, se debe generar una malla con suficiente resolución cerca de estas paredes ya que se debe tener en cuenta que, aunque este tipo de zonas sólo suponen una pequeña fracción sobre el total del dominio computacional, de su correcta resolu-

ción dependerá la calidad de las simulaciones aerodinámicas llevadas a cabo. Sin embargo, la generación de una malla que resulte adecuada para las necesidades expuestas no es una tarea sencilla: la complejidad de los detalles geométricos presentes en la mayoría de los modelos, tales como esquinas cóncavas/convexas, micro-superficies, etc., pueden provocar no sólo fallos en la propia generación de la malla, sino que pueden conducir al empleo de enormes tiempos, y por tanto de sus costes laborales asociados, destinados a este preproceso. Por tanto, en el entorno industrial, se puede afirmar que la generación de mallas para simulaciones de flujos viscosos aún constituye un importante desafío. El presente trabajo presenta un nuevo, robusto y eficiente método para la generación de mallas viscosas de forma que puedan ser empleadas para el análisis de CFD de complejas configuraciones tridimensionales. Conceptualmente el método aquí propuesto se englobaría dentro de las estrategias conocidas como de mallado híbrido. En primer lugar, el método desarrolla un robusto criterio para la correcta validación de la malla generada en problemas complejos, para, a continuación implementar una nuevo esquema de validación de la geometría que permite reducir significativamente el número de nodos a chequear durante los sucesivos procesos de inserción que se producen en todo algoritmo de generación de mallas. La principal contribución, por tanto, del método aquí propuesto es la de reducir sustancialmente el tiempo requerido para la generación de mallas respecto a los códigos comerciales y libres disponibles hasta la fecha, ofreciendo, al mismo tiempo, unos niveles de robustez iguales o superiores a los mismos.

La precisión de la predicción de la temperatura es de suma importancia para el análisis de turbinas. Como tiene un impacto directo para asegurar la seguridad de la turbina y lograr una operación razonable de larga duración, mientras tanto, aumentar la eficiencia de la turbina evitando el uso de aire de refrigeración excesivo descargado de las etapas del compresor. Tradicionalmente, los análisis de CFD y de conducción térmica sólida se han realizado por separado en diferentes etapas del ciclo de diseño. Varias hipótesis están implícitas en este proceso y la exactitud de este tipo de métodos se basa inevitablemente en la base de datos empírica y el juicio de ingeniería. Por lo tanto, en casos de flujo altamente complejos tales como flujos de cavidades de turbina o problemas de enfriamiento interno, los métodos tradicionales pueden conducir a errores con-

siderables en la estimación del campo de temperatura. Los métodos de transferencia de calor fluido-sólido acoplados, en los que los problemas de transferencia de calor fluido y sólido se resuelven simultáneamente sin el uso de otras hipótesis, se garantiza una mayor precisión debido a que el flujo de calor y la continuidad de temperatura a través de la interfaz fluido-metal se garantizan por construcción. Debido a su relevancia, los algoritmos de transferencia de calor acoplados han recibido considerable atención desde diferentes perspectivas de ingeniería en las últimas décadas. Sin embargo, la principal desventaja de los métodos de transferencia de calor acoplados son los tiempos de cálculo relativamente grandes comparados con el de una simulación de CFD independiente desacoplada. Por lo tanto, es difícil de aplicar en aplicaciones prácticas de ingeniería. Esta tesis también propone un nuevo, rápido y robusto método de transferencia de fluido-sólido de acoplamiento suave basado en la mejora de un enfoque de acoplamiento de Dirichlet-Robin para aumentar la precisión del campo de temperatura. Este método ha sido validado y aplicado a casos representativos de turbomáquinas.

Los flujos giratorios pueden ser satisfechos en los motores aero debido a la rotación del sistema. Los flujos rotatorios bajo Coriolis y las fuerzas centrífugas que afectan los patrones de flujo drásticamente en comparación con los casos estacionarios. Es difícil llevar a cabo un diseño real sin una buena comprensión de la física de flujo rotatorio. Por último, este trabajo proporciona la comprensión del efecto de rotación en los canales de refrigeración internos de la pala de rotor de la turbina. También se ha estudiado el comportamiento de flujo y transferencia de calor de un canal giratorio representativo del motor.

Palabras clave: generación de malla viscosa, malla híbrida, método de transferencia de calor acoplado, efecto de rotación, refrigeración interna, transferencia de calor

Contents

List of Figures	ix
List of Tables	xi
List of Algorithms	xiii
Nomenclature	xx
1 Introduction	1
1.1 Challenges in Numerical Methods for Turbine Internal Cooling Problems . . .	3
1.1.1 Three-Dimensional Numerical Modelling	3
1.1.2 Accurate Temperature Prediction	5
1.1.3 Rotational Effects in Engine Similar Configuration	6
1.1.4 Numerical Issues	7
1.2 Objective and Outline	8
2 Hybrid Viscous Grid Generation	11
2.1 Introduction	12
2.2 Review of Boundary Layer Mesh Generation Techniques	16
2.2.1 Advancing Layer Methods	16
2.2.2 Surface Normal Calculation	17
2.2.3 Numerical Treatment of Surface Normals	21
2.3 Surface Mesh Generation	23
2.4 Viscous Mesh Generation	24
2.4.1 Face Identifications	26
2.4.2 Node Normal Spacing Specification	28

2.4.3	Determination of Marching Direction	29
2.4.4	New Layer Generation	32
2.4.5	Additional Treatments	38
2.5	Accelerating Method	39
2.5.1	The OLB Checking Algorithm	40
2.5.2	Algorithm Complexity Analysis	44
2.6	Volumetric Mesh Generation	46
2.7	Test Cases	46
2.7.1	Turbine Leading Edge Impingement Cooling Mock-up	46
2.7.2	NASA Common Research Model	49
2.7.3	Internal Cooling Channels of Turbine Blade	51
2.8	Summary	55

3 Coupled Fluid-Solid Heat Transfer Method 57

3.1	Introduction	58
3.1.1	Background	58
3.1.2	Coupled heat transfer methods	61
3.2	Baseline Loosely Coupled Strategy	67
3.3	Analysis of Loosely Coupled Approach with Dirichlet-Robin Interface	70
3.3.1	Fully Converged CFD	71
3.3.2	Incompletely Converged CFD	73
3.3.3	Discussion on the Convergence Level	77
3.3.4	Implementation Details	79
3.3.5	Validation	81
3.4	Practical Case Study	83
3.4.1	Leading Edge Impingement Cooling Channel	83
3.4.2	Turbine Stator Well Cavity	84
3.4.3	Computational Test Matrix	85

3.5	Results and Discussion	86
3.5.1	Convergence Behaviour	88
3.5.2	Computational Time	90
3.6	Summary	93
4	Rotational Effect of Internal Channel Flows	95
4.1	Navier-Stokes Equations in Relative Frame of Reference	96
4.2	Relevant Non-Dimensional Parameters in Rotating Channels	97
4.2.1	The reduced Static Pressure	98
4.2.2	The Boussinesq Approximation	99
4.2.3	Non-dimensional Form	100
4.3	Rotational Effects in Internal Cooling Channel	102
4.3.1	Flow in Two-dimensional Rotating Channels	103
4.3.2	Flow in Three-dimensional Rotating Channels	106
4.3.3	Coriolis Effects on the Stability of Shear Layer	111
4.3.4	Rotating Buoyancy Effects	114
4.3.5	Heat Transfer in 3D Rotating Channels	115
4.4	Flow and Heat Transfer in 3D Ribbed Channels	117
4.4.1	Rotational Effects on the Inter-rib Flow	118
4.4.2	Secondary Flow Contours	118
4.4.3	Heat Transfer Performance	120
4.5	Summary	122
5	Uneven Wall Heating Effect of an Engine Representative Cooling Channel	123
5.1	Introduction	124
5.2	Rotating Channel Model	126
5.3	Experimental Set-up	127
5.4	Numerical Set-up	130
5.5	Results	132
5.5.1	Segment Averaged Heat Transfer Validation	133

5.5.2	Uneven Wall Heating Effects on Heat Transfer Distribution	135
5.5.3	Evolution of Overall Heat Transfer Performance	137
5.5.4	Secondary Flow Characteristics	140
5.5.5	Inter-Rib Flow Characteristics	143
5.5.6	Main Stream Flow	147
5.6	Summary	149

6	Conclusions and Future Work	151
----------	------------------------------------	------------

Bibliography	155
---------------------	------------

List of Figures

1.1	Gas Turbines Thermal Trend (From Dr. Toyooki Yoshida, National Aerospace Laboratory, Japan)	2
1.2	Typical high pressure turbine blade internal cooling system [1]	4
2.1	Different types of meshes for viscous flow simulations for a two-dimensional high-lift wing airfoils	14
2.2	Deflection of node normal due to non-uniform surface mesh at a 3D convex corner	18
2.3	Sketch of the visibility cone and the normal cone at a 2D convex corner	20
2.4	Two optimal normal calculation methods based on the equal visibility criteria proposed by (a): Pirzadeh [2] and (b): Aubry and Lohner [3]	20
2.5	Different normal treatments at convex ridges: (a) non-smoothing; (b) with Laplacian smoothing; (c) multiple normal approach	22
2.6	Different normal treatments at concave corners: (a) non-smoothing, (b) with Laplacian smoothing, (c) direction enrichment	22
2.7	Surface mesh of a leading edge impingement cooling channel with film cooling holes	25
2.8	Flow chart of viscous mesh generation	26
2.9	Identification of faces and edges	27
2.10	Thickness of each viscous layer controlled by the growth factor β (left: $\beta > 1$, right: $\beta \approx 1$,)	28
2.11	Normal treatments at non-viscous edges	30

2.12 Comparison of layer thickness correction: (a) without correction; (b) with correction	31
2.13 2D (a) and 3D (b) boundary layer meshes with exposed surfaces	33
2.14 Sketch of the minimum requirements of the neighbouring cells to form a prism in the target cell	34
2.15 Intersection problems for neighbouring elements: (a) intersection within the same front, (b) intersection with a different front	35
2.16 Sketch of proximity and intersection checking procedure	36
2.17 Sketch of periodic faces	39
2.18 Pre-layer checking, illustrated in 2D	40
2.19 Illustration of the outcome of the outer layer boundary checking pre-processing step for a ribbed cooling passage. Red: cells which passed the test. Cyan: cells which failed to pass the test.	41
2.20 Model and final mesh of impingement cooling channel	47
2.21 Plane-cut of the final mesh for the impingement cooling channel.	48
2.22 Quality comparison. Mesh A: constant thickness with smooth transition (left), Mesh B: reduced thickness height (15%) at the narrow regions (right).	48
2.23 Accelerating factor comparison between “Skyline” and <i>baseline</i> . $\alpha = (1 - \text{OLB success rate})$	50
2.24 Mesh example of Common Research Model (CRM)	51
2.25 Surface mesh of three internal cooling channels of rotor blade	52
2.26 Mesh cut-off of three internal cooling channels of rotor blade	53
2.27 Temperature field of internal cooling channels of a turbine blade	54
2.28 Overall heat transfer comparison for leading edge cooling channel: CFD vs EXP	54
3.1 Development of high pressure turbine material versus turbine inlet temperature [4]	58

3.2	Schematics of the heat flux as a function of wall temperature when the h is of interest: (a) linear function, h is independence of T_w (b) nonlinear function, h is variant with T_w	59
3.3	One-dimensional coupled fluid/solid heat transfer problem	70
3.4	Dirichlet-Robin approach convergence sketch (left) and stability region (right)	73
3.5	Error sequence of the incompletely converged CFD scheme. Inset: Converge history for different CFD strategies.	76
3.6	Scheme of dynamic virtual HTC approach	79
3.7	Computational domain and mesh for the coupled heat transfer problem of laminar air flow over a finite thickness flat plate	81
3.8	Coupled wall temperature distribution along a flat plate for different k_s/k_f ratios. Inset: Span-wise temperature distribution at the flat plate mid-point	82
3.9	Leading edge impingement cooling channel configuration for coupled heat transfer analysis	84
3.10	Left: Stator well cavity configuration for coupled heat transfer analysis. Right: Computational domain and measurement points	85
3.11	Impingement cooling channel coupling configuration wall temperature, in K , (left) and HTC, in mW/mm^2K , (right) distributions	86
3.12	Stator well cavity configuration coupling surface temperature in K (left) and HTC in mW/mm^2K (right) distributions	87
3.13	Relative error of coupled heat transfer method and uncoupled method at measurement points	87
3.14	Convergence history for the impingement cooling channel configuration	88
3.15	Convergence history for the stator well cavity configuration	89
3.16	Convergence history as a function of the computational time for the impingement cooling channel configuration. Every individual symbol represents an iteration	90

3.17	Convergence history as a function of the computational time for the stator well cavity configuration. Every individual symbol represents an iteration	91
4.1	Diagram of rotating components in a modern aero-engine	98
4.2	Schematic model of internal cooling channel in turbine rotor blade	102
4.3	Stream-wise flow profile in a 2D channel under rotation	105
4.4	Stream-wise pressure profile in a 2D channel under rotation	106
4.5	Stream-wise velocity field and secondary flow contours in a 3D smooth channel under rotation $Re = 21000$	107
4.6	Stream-wise flow profile in a 3D smooth channel under rotation at $Re = 21000$	107
4.7	Analytical solution of Stream-wise and vertical velocity profiles in Ekman-like layer (Equation 4.28)	110
4.8	Comparison of analytical solution and computational results of Stream-wise and vertical velocity profiles in Ekman-like layer for different Ekman numbers in 3D channels	111
4.9	Reduced pressure field and secondary flow contours in a 3D smooth channel under rotation ($Ro = 0.12$)	112
4.10	Coriolis forces on fluid particles in a rotating channel	113
4.11	The rotating buoyancy effect on the stream-wise flow profile in a 3D smooth channel	115
4.12	Rotational effect on heat transfer performance for a 3D rotating channel as a function of Rotation numbers	116
4.13	3D ribbed channel model	117
4.14	Streamline and stream-wise velocity contours of inter-rib flow under different rotating conditions at $Re = 15000$	119
4.15	Secondary flow in one-side ribbed channel under different rotating conditions at $Re = 15000$	119
4.16	Heat transfer performance of one-side ribbed channel under different rotating conditions at $Re = 15000$	120

4.17	Comparison of sectional averaged Nu/Nu_0 between numerical simulation and experimental data as a function of Rotation numbers	121
5.1	Cooling channel sketch	127
5.2	Main test section in experimental set-up	128
5.3	Hybrid mesh of computational domain	132
5.4	Numerical model and boundary conditions	133
5.5	Segment index notation	133
5.6	Comparison of segment averaged Nu/Nu_0 of the first passage at nominal condition ($Re = 15000$, $Ro = 0.3$ and $Bo = 0.17$), with the ribbed wall heated . . .	135
5.7	Numerical results of Nu/Nu_0 distribution for two different wall heating conditions at nominal condition ($Re = 15000$, $Ro = 0.3$ and $Bo = 0.17$)	136
5.8	Evolution of overall Nu/Nu_0 for uneven heat conditions as a function of the Buoyancy and Rotation numbers	136
5.9	Effect of uneven heating under rotation as a function of the Buoyancy number Solid symbols: all walls heated; hollow symbols: rib wall heated	138
5.10	Velocity magnitude and secondary flow patterns at channel inlet	141
5.11	Velocity magnitude and secondary flow patterns (arrows) at the middle of segment no. -3 for uneven wall heating conditions under different Buoyancy numbers	142
5.12	Stream-wise velocity contours and streamlines in inter-rib region for different wall heating conditions under different Buoyancy numbers	145
5.13	Span-wise velocity contours in inter-rib region for different wall heating conditions under different Buoyancy numbers	146
5.14	Schematic model of flow in ribbed trailing wall tests, under Coriolis and buoyancy forces	148
5.15	Stream-wise velocity profile along channel height at middle plane of segment -3 for different wall heating conditions under different Buoyancy numbers . . .	148

List of Tables

2.1	Grid size and wall-clock generation time of the impingement cooling channel.	49
2.2	Comparison of the execution time between the <i>baseline</i> and proposed algorithm for the meshes A and B of the impingement cooling configuration.	49
2.3	Grid size and wall-clock generation time of three different mesh generation tools for the CRM model.	51
2.4	Grid size and wall-clock generation time of three internal cooling channels of rotor blade	52
3.1	Test matrix for coupled heat transfer analysis	85
5.1	Mesh sensitivity study	131
5.2	CFD test matrix	131
5.3	Geometry parameters of one-wall ribbed channels	137
5.4	Test parameters of rotating channels	139

List of Algorithms

- 2.1 Surface mesh generation scheme by means of Steiner triangulations [5] 24
- 2.2 Iterative weighted normal calculation 29
- 2.3 Validity checking of prismatic elements 33
- 2.4 Compatibility checking of neighbour elements 35
- 2.5 Proximity and intersection checking procedure 36
- 2.6 OLB checking algorithm 42
- 2.7 Layer generation process of OLB method 43

Nomenclature

Abbreviations

3D	Three-dimensional
ADT	Alternating digital tree
AR	Aspect ratio
CAD	Computer-aided design
CFD	Computational fluid dynamics
DNS	Direct numerical simulation
HPT	High-Pressure Turbine
LES	Large eddy simulation
MGG	Mapped space Generated Grid
MUSCL	Monotonic Upstream-Centered Scheme for Conservation Laws
NURBS	Non-uniform rational basis spline
OLB	Outer Layer Boundary
RANS	Reynolds-averaged Navier–Stokes
SOR	Successive over-relaxation

TIT Turbine inlet temperature

Greek Symbols

α Angle; OLB factor; thermal diffusivity

β relaxation parameter; coefficient of thermal expansion

δ Thickness

ε Error residual

η Non-dimensional length

γ heat capacity ratio

μ Dynamic viscosity

ν Kinematic viscosity

Ω Angular velocity; flow domain; control volume

ω Specific dissipation; vorticity

Φ Viscous dissipation term

ρ Density

τ Non-dimensional time; viscous stress tensor

ε Convergence level

Roman Symbols

A Area

e Unit vector

F	Inviscid and viscous fluxes
q	heat flux
R	Vector of conservative variables
r	Position vector
v	Absolute velocity
w	Relative velocity
<i>a</i>	Speed of sound
<i>Bi</i>	Biot number
<i>Bo</i>	Buoyancy number
<i>C</i>	Cell, heat capacity
<i>D</i>	Artificial dissipation
<i>D</i>	Fourier number
<i>D_h</i>	Hydraulic diameter
<i>E</i>	Element
<i>e</i>	Error
<i>e</i>	rib height
<i>Ec</i>	Eckert number
<i>Ek</i>	Ekman number
<i>G</i>	Amplification factor
<i>Gr</i>	Grashof number

H	non-dimensional distance
h	heat transfer coefficient; enthalpy
I	rothalpy
i	Internal energy
k	Turbulent kinetic energy; thermal conductivity
L	characteristic length
m	Mass flow rate
Ma	Mach Number
N	Normal; node
Nu	Nusselt number
P	Block-Jacobi preconditioning matrix
p	Point; pressure
p	pressure
Pr	Prandtl number
Q	Volumetric heat sources
R	Residual; the gas constant
Re	Reynolds number
Ro	Rotation Number
S	Boundary; source term

T	Temperature
t	time
w	Weight parameter; relative velocity
y	Spacing in normal direction; dimensionless wall distance

Subscripts

$*$	Virtual value
0	Initial value
∞	ambient condition
cen	Centrifugal
cor	Coriolis
Ex	External
f	Fluid
i	Index; inertial frame
i	Inertial frame
in	inlet
int	intermediate
L	Reference length
m	mean value
n	Iteration number

<i>out</i>	outlet
<i>p</i>	Point
<i>r</i>	relative frame, radial
<i>red</i>	Reduced
<i>ref</i>	Reference value
<i>rel</i>	Relative
<i>s</i>	Solid
<i>T</i>	Temperature
<i>t</i>	Total or stagnation
<i>w</i>	Wall

1 Introduction

Contents

1.1	Challenges in Numerical Methods for Turbine Internal Cooling Problems	3
1.2	Objective and Outline	8

High-Pressure Turbines (HPTs) of modern aero-engines operate at very high temperatures to improve thermal efficiency and increase power. Figure 1.1 illustrates the increasing turbine inlet temperature in recent advanced gas turbines. Nowadays, such high temperatures (higher than 1700°C) have already been far in excess of typical metal material melting temperatures (around 1000°C). Turbine rotor blades, as well as all of the gas swept parts, are subject to the engine's most aggressive heat loads since the supplied working fluid at this stage is at the peak cycle temperature, and the work generation process in the turbine accelerates the flow to high speeds, which in turn results in enormous heat fluxes on the component surfaces. In addition to the development of new high temperature capability materials, and thermal coatings technologies, highly sophisticated cooling techniques are essential to maintain gas turbines operability and durability under such extreme working conditions.

Gas turbine cooling techniques can be either internal or external. Internal cooling is achieved by passing the coolant air through several cooling passages which can be enhanced by different shapes of turbulence promoters inside the blade to extract the heat from the blade metals. Jet impingement cooling and pin fin cooling may also be used as a method of internal cooling at certain locations of turbine blades.

Figure 1.1: Gas Turbines Thermal Trend (From Dr. Toyoaki Yoshida, National Aerospace Laboratory, Japan)

External cooling is also known as film cooling, coolant air is ejected out from internal cooling passages through array of cooling holes or slots forming a coolant film layer to protect the external surface of the airfoils from the main stream hot gas path. More details of general cooling techniques can be found in a review book from Han et al. [6].

In a cooling system, the coolant air is extracted from compressor stages of the engine. Too little bleeding air results in higher blade temperatures and reduced turbine component life; in contrast, too much extraction incurs in a severe penalty in the thermal efficiency. Moreover, subject to complex geometries and rotating buoyancy interactions, the cooling system invariably results in highly turbulent flows whose individual features interact in very complex ways. Due to these complications, the designing of turbine cooling system plays an extremely important and very challenging role in aero-engine development. However, the heat transfer and cooling processes are at the limit of current understanding and engine designers rely heavily on experiments, empirical tools and engineering judgement to produce new designs.

1.1 Challenges in Numerical Methods for Turbine Internal Cooling Problems

With the rapid growth of computer power, high fidelity numerical methods have been used in internal cooling analysis. Numerical methods are quite valuable in identifying the complex underlying physics of cooling flows and to provide more details in the flow-field and heat transfer phenomena than experimental methods. In addition, it allows the designer to test and optimise a wide range of conditions, at a relatively low cost, as sometimes at operating conditions experimental techniques are either not feasible or have a limited accuracy. Therefore, in many cases, numerical methods is the only way to understand the complex flow physics systematically. However, due to the complexity of some characteristics such as geometries, rotating flow physics and heat transfer behaviours, the application of numerical methods in practical internal cooling problems has encountered a number of challenges which need to be addressed.

1.1.1 Three-Dimensional Numerical Modelling

Mesh generation is a critical process in Computational Fluid Dynamics (CFD) since mesh quality not only has a direct impact on the final solution, but also becomes a bottleneck of the whole analysis process. Grid spacing should be smoothly and sufficiently refined in certain regions to properly resolve steep solution gradients, whereas it should be coarser in other regions not compromised by mesh resolution considerations. A key element to achieve good quality meshes in viscous flow simulations at high Reynolds numbers is a high enough mesh resolution near solid walls to properly capture boundary layer gradients in the wall-normal direction. These regions account only for a small fraction of the computational domain but are directly linked with the quality of aerodynamic simulations. However, significant difficulties arise in the presence of complex geometric features such as multiple concave/convex corners, micro-surfaces, etc., which may result in mesh generation failure, or considerable elapsed time of the mesh generation process with the subsequent associated labour cost.

Figure 1.2 shows a typical high pressure turbine rotor blade geometry consisting of internal

Figure 1.2: Typical high pressure turbine blade internal cooling system [1]

passages fed by cooling air at the base through the rotor hub. It can be seen that, the internal cooling passages are composed by various kinds of complex multi-pass channels equipped with different sorts of turbulent promoters or film cooling tubes to enhance convective heat transfer performance. Due to this complexity, the discretization of the computational domain using mesh generation algorithms remains one of the most difficult tasks in internal cooling applications.

Firstly, there is a critical problem associated to the source of the geometry itself, and its import process into the meshing system. In a design or analysis loop, the internal cooling models for mesh generation are generally used or translated directly from realistic three dimensional CAD models. The geometry exported from a CAD system is almost always ‘dirty’ due to numerical tolerance issues within the CAD system (often exacerbated by greatly differing scales within the geometry) and lack of numerical compatibility between various geometrical representations, ruled surfaces, NURBS patches, surfaces of revolution, etc [7]. Such models consist of hundreds to thousands of complex warped faces, which have a large size disparity, with degenerated or badly parametrised entities. These complexities in the grid generation process can make the generation of the mesh infeasible, or be associated with non-affordable labour costs or elapsed time.

Secondly, fluid flows in internal cooling passages are very complex and highly turbulent, there-

fore very fine meshes are required. However, due to the compromised consideration of the mesh size, highly intensive refined meshes may only allow to be used near concave/convex ridges and turbulence promotion entities regions. Therefore, the surface mesh is highly non-uniform. The most common validity checking algorithms may fail for elements with such large disparity in size [8]. In addition, the reference length used for near wall mesh thickness estimates is characterised by the hydraulic diameter of the cooling passages, which is much smaller than the length of the whole passage. As a consequence, in order to have a high resolution viscous mesh to capture turbulent flow and heat transfer effects, the thicknesses of viscous cells must be small. Therefore, compared to the characteristic size of surface background cells, these viscous cells are extremely stretched (It is easy to find cell aspect ratios of $O(10^3 \sim 10^5)$), requiring very robust mesh generation methods.

Thirdly, with rapid growth of computer power, nowadays, it is possible to perform steady RANS simulation using a meshes with $10 \sim 100$ million nodes in acceptable times. However, the meshing process for a complex 3D model with such a large mesh size is usually very time-consuming. Even for experienced engineers, it may take days to weeks to create a mesh for a very complex internal cooling geometry [9, 10]. When considering extra uncertainties coming from badly parametrised entities in practical applications, non watertight models, etc, a great waste of engineering hours and long meshing turn around time are frequent. This problem becomes more critical when many meshes are needed in a sequential process, such as shape optimisation in cooling system design. Therefore both robustness and efficiency are highly concerned in meshing internal cooling geometries.

1.1.2 Accurate Temperature Prediction

As the turbine inlet temperature increases, the heat transferred to the turbine blade also increases. The level and variation in such high temperature have a direct impact on turbine material, which causes thermal stresses and blade failures, and must be limited to ensure safety and achieve reasonable long-life operation. It has been shown that, the blade life maybe reduced by half if the blade metal temperature prediction is off by only 30°C [11]. In addition, around 20%

to 30% [12] of the core flow is purged for the turbine blade cooling system, which creates a considerable degradation of the gas turbine cycle efficiency. Therefore, the accuracy of temperature prediction is of paramount importance for turbine analysis, since an accurate prediction can prevent local hot spots and increase turbine blade life, avoiding at the same time the use of excessive cooling air and therefore allowing a higher turbine inlet temperature.

Traditionally, CFD and solid thermal conduction analyses have been performed separately at different stages of the design loop. The fluid solver provides a simplified estimate of the thermal characteristics at the wall interfaces, usually under the form of a heat transfer coefficient. Then this information is applied to solid thermal codes for metal temperature calculation, which in turn is further used to predict thermal stresses and displacements. Several hypotheses are implicit in this process and the accuracy of this kind of methods inevitably relies on empirical database and engineering judgement. Therefore, in highly complex flow cases such as turbine main annulus and cavity flows or internal cooling problems, traditional methods may lead to considerable errors in the estimate of the temperature field, and to severe logistic problems to handle the exchange of information among different codes.

Greater accuracy is offered by coupled fluid-solid heat transfer methods, in which fluid and solid heat transfer problems are solved without the use of further hypotheses, since the heat flux and temperature continuity across fluid-metal interface is ensured by construction. Because of their relevance, coupled heat transfer algorithms have received considerable attention from different engineering perspectives in the last decades [13, 14, 15, 16, 17]. However, the main disadvantage of coupled heat transfer methods is the relatively large computing time compared to that of an uncoupled stand-alone CFD simulation. Therefore, they have traditionally been difficult to apply in realistic engine representative applications.

1.1.3 Rotational Effects in Engine Similar Configuration

Turbine rotor blades rotate at very high rotational speed. Coolant flow passing through the internal cooling channel experiences two additional forces: Coriolis forces and centrifugal buoyancy forces. Subject to rotation, Coriolis forces induce secondary flows and vortexes in the

transverse direction along the channel. Therefore, the flow and heat transfer patterns are very much different from those in stationary channels.

Although tremendous experimental campaigns have been carried out in last fifty years to set-up databases for design purposes, most of the studies provided only heat transfer data and lacked a deep understanding and proper flow-field analysis, because of the difficulties associated with the velocity measurement under rotating conditions, which is still a challenge task even for today [18].

Additionally, in real turbine blades, the flow in internal cooling systems depends on combined geometric parameters which characterise the shape and orientation of the cooling passage, the shape and arrangement of turbulent promoters, and flow conditions. The flow promoted by these geometrical features, combined with the flow induced by rotational effects results in a extremely complex 3D flows with large diversity of secondary flows, which could be far more complex than that considered in general laboratory experiments. Thus, it is difficult to carry out real designs without a good understanding of the physics of rotating flows.

Therefore it is necessary to conduct a detailed examination to numerically understand the importance of the flow-field and the heat transfer associated with internal cooling and perform a systematic analysis of the relevance of each non-dimensional parameter in engine similar configurations.

1.1.4 Numerical Issues

In addition to the problems mentioned above, the characteristics of cooling flow and heat transfer behaviours, can result in difficulties to the numerical scheme itself. For instance, the flow velocities in internal cooling channels are in most cases in the very low Mach number range ($Ma < 0.1$) to avoid large pressure losses due to compressibility effects. Therefore, for widely used time-marching density-based schemes, when the flow velocity is small, compared to the acoustic wave-speed, the time-dependent equation system becomes stiff and solvers converge slowly [19]. Time-marching procedures may suffer severe stability and accuracy restrictions

and become inefficient. This effect of system stiffness on solution convergence is well known, for both explicit and implicit schemes [20]. Therefore, preconditioning schemes have to be implemented.

Another key issue is about the turbulence modelling. Because Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES) are still limited to simple geometries or relatively small domain investigations, Reynolds-averaged Navier Stokes (RANS) with proper turbulence models are still the most important workhorse for fluid and heat transfer predictions in engineering applications. Rotational effect and a variety of turbulent promoting devices create highly complex 3D flow with multiple separation-reattaching patterns and large disparity of flow re-circulations. It induces complex flow mixing as opposed to simple shear flows such as a conventional flat plate or zero pressure gradient boundary layer. The anisotropy Reynolds stress terms may play an important role, leading to a bad prediction in flow and heat transfer for conventional turbulence models. Furthermore, rotational terms cause redistribution of turbulence energy among its components and affect the Reynolds stresses modifying the mean flow implicitly. In addition, Coriolis terms explicitly enter the mean momentum equations, thus the effect of rotation on turbulent energy may be quite dramatic, although the Coriolis force, does not enter directly in the energy equation [21]. Therefore, advanced turbulence modelling is usually required.

1.2 Objective and Outline

The objective of this thesis is to develop numerical techniques relevant to the study of turbine internal cooling simulations for practical engineering applications. It aims to improve the methodology and understanding of internal cooling problems in three aspects:

- Development of robust, efficient 3D mesh generation methods for complex configurations;
- Development of a time efficient coupled fluid-solid heat transfer method;

- Understanding of rotational effect and its impact in engine relevant configurations.

This thesis is organised in following manner to cover the aforementioned objectives:

Chapter 1 presents the overview about the internal cooling of the gas turbine blade including a brief discussion on the difficulties encountered in the numerical methods.

Chapter 2 describes the development and implementation of a robust 3D boundary mesh generation method for complex internal cooling configurations for flow field investigation. Engine representative configurations are used to explore the robustness and efficiency of the algorithm. The presented algorithm is proved to be more efficiently than common commercial or open source codes with equal or higher robustness.

Chapter 3 proposes an efficient loosely coupled fluid-solid heat transfer method to minimise the computational overhead of using coupled heat transfer methods. The basic theoretical analysis is presented. The comparisons of computational time cost between our algorithm against standard algorithm and CFD alone simulations are also presented with two turbine representative 3D cases.

Chapter 4 introduces and studies the general rotational effect on flow and heat transfer behaviour in rotating channels.

Chapter 5 presents an example of rotational effect studies on flow and heat transfer in an engine representative internal cooling channel of turbine rotor blade.

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

2 Hybrid Viscous Grid Generation

Contents

2.1	Introduction	12
2.2	Review of Boundary Layer Mesh Generation Techniques	16
2.3	Surface Mesh Generation	23
2.4	Viscous Mesh Generation	24
2.5	Accelerating Method	39
2.6	Volumetric Mesh Generation	46
2.7	Test Cases	46
2.8	Summary	55

This chapter describes the development and implementation of an efficient and robust advancing layer viscous mesh generation method for 3D complex configurations. The final mesh is created in a prismatic-tetrahedral hybrid manner for Computational Fluid Dynamics (CFD) calculations. The implementation of the boundary mesh generation method with enhanced validity checking is described in details. Then, a new approach is proposed to improve the efficiency of the baseline algorithm. The accelerated method employs a pre-processing algorithm to identify cells which can be generated without further checking, whereas the rest of the cells are checked by a standard layer-by-layer procedure. This method significantly reduces the number of proximity/intersection checking operations, substantially decreasing the mesh generation wall-clock time while maintaining the same robustness and quality of the baseline method.

2.1 Introduction

Computational Fluid Dynamics (CFD) has demonstrated its capability to predict complex flow field and heat transfer behaviour via numerical methods in internal cooling channels of gas turbine rotor blades in the past decades. In many cases, CFD may be the only way to understand the complex flow physics in internal cooling problems due to the difficulties to measure flow field in realistic engine or engine-similar experiments.

Mesh generation is an essential and critical process in CFD, since the mesh quality can have a direct impact on the accuracy of CFD solution. The generated mesh must be dense enough so that the numerical approximation is accurate, but it can not be so dense that the solution is impractical to obtain. In addition, grid spacing should be smoothly and sufficiently refined in certain regions to properly resolve steep solution gradients, whereas it should be coarser in other regions compromised by affordable mesh-size. Therefore, three-dimensional mesh generation is still a bottleneck problem in CFD, especially when dealing with extremely complex geometries. For instance, in a real gas turbine blade, the internal cooling passages are usually composed by various kinds of complex multi-pass channels equipped with different kinds of turbulent promoters or film cooling tubes to enhance heat transfer performance. Due to this kind of complex geometry issues, the generation of numerical grids remains one of the most difficult tasks in internal cooling applications.

The difficulties resulting by the geometry issues can be further broken down as follows:

Firstly, a critical associated problem is the source of the geometry itself and its import into the meshing system. In a design or analysis loop, the internal cooling models for mesh generation are generally used or translated directly from realistic three-dimensional CAD models. The geometry exported from a CAD system is almost always “dirty” due to numerical tolerance issues within the CAD system (often exacerbated by greatly differing scales within the geometry) and lack of numerical compatibility between various geometrical representations-ruled surfaces, NURBS patches, surfaces of revolution, etc [7]. Such models consist of hundreds to thousands of complex warped faces, with a large size disparity, with degenerated or badly para-

metrised entities. These complexities in the grid generation process can make the generation of the mesh infeasible, or at non-affordable labour costs or elapsed times.

Secondly, flows in internal cooling systems are very complex and highly turbulent, therefore very fine meshes are required. However, due to the compromised consideration of mesh size in time-affordable CFD simulations, highly intensive refined meshes may only allow to be used at certain regions, for instance at shear layers or highly separated regions near ridges and turbulence promotion devices. Therefore, the surface mesh can be highly non-uniform. The common validity checking may fail when treating elements with such large disparity in size [8]. In addition, the reference length used for near wall mesh thickness estimation is characterised by the hydraulic diameter of the cooling passages, which is much smaller than the length of the passage. In order to have a high resolution viscous mesh to capture turbulent flow and heat transfer effects, the cell spacing normal to the wall has to be very small. Thus, comparing to the characteristic size of surface background cells, these viscous cells are extremely stretched ($O(10^3 \sim 10^5)$), requiring a very robust method to ensure the validity.

Thirdly, with rapid growth of computer power, nowadays, it is possible to perform steady RANS simulation using meshes with $O(10^7 \sim 10^8)$ nodes level in affordable times. However, the meshing process for a complex 3D model with such a large mesh size is usually very time-consuming. It may take days to weeks to create a mesh for a very complex internal cooling geometry even for experienced engineers[9, 10]. When considering extra uncertainties coming from badly parameterized entities in practical applications, non watertight models, etc, a great waste of human-work and meshing turn around time is frequent. This problem becomes more critical when many meshes are needed in a sequential process, such as shape optimisation in cooling system design. Therefore, both robustness and efficiency are highly concerned in meshing internal cooling geometries. In addition, the mesh generator should have the capability with less man-intervention, in order to be feasible for non-experienced users or sequential mesh generation.

There are variety ways to create computational mesh for CFD simulations. Figure 2.1 shows an two-dimensional high-lift wing airfoils with different type of meshes commonly used for

(a) Multiblock structured mesh (b) Unstructured cartesian mesh

(c) Hybrid Mesh

Figure 2.1: Different types of meshes for viscous flow simulations for a two-dimensional high-lift wing airfoils

viscous flow simulations. Structured meshes (curvilinear meshes) are generally preferred in CFD simulations because of their high regularity, computational efficiency and fast generations. However, due to the geometrical complications mentioned above, it is almost impossible to apply in realistic internal cooling studies, at least without a heavy user intervention. Although, this problem can be overcome by means of multi-block structured mesh [22], in which the computational domain is sub-divided into multiple blocks (see Figure 2.1 (a)), allowing much more geometrical flexibility. This process is usually very time-consuming and even for experienced engineers, it may take days to weeks to create mesh for a very complex internal cooling geometry [23]. Therefore, an unstructured based, automatic and less user-demanding method is preferable.

Automatic octree-based Cartesian mesh [22] could be an alternative option, because of its capability to mesh for complex domains while maintaining the advantage of creating tetrahedral elements for CFD simulations. However, the treatment of walls sometimes becomes a problem to perform accurate simulations [24]. Several methods have been proposed to resolve these issues geometrically and computationally, for instance the cut-cell method [25], the building-

cube method [26] and the immersed boundary method [27]. In addition, extra difficulties may be encountered when inserting good quality boundary layer meshes. Additionally, Cartesian meshes usually requires the support for over-set grids or hanging nodes, which is not necessarily available in all CFD solvers for industrial use. So that there is still room for further improvements.

On the other hand, three-dimensional advancing front [28] or Delaunay [29, 30] based triangular and tetrahedral mesh are widely used because of their flexibility for complex geometries and capability for local refinements. Nowadays, these two methods have been developed drastically and proved to be very mature to handle complex geometries and to refine grids in certain regions [31]. However, classical triangular or tetrahedral elements are usually used to generated isotropic grids, which are only suitable for inviscid Euler flow simulations. High quality hybrid or anisotropic tetrahedral elements are required near the walls to capture the viscous flow and heat transfer behaviours (see Figure 2.1 (c)).

As discussed above, a key element to achieve good quality meshes in viscous flow simulations at high Reynolds numbers is to generate high enough mesh resolution near solid walls to properly capture boundary layer gradients in the wall-normal direction. These regions account only for a small fraction of the computational domain but may contains 40%-70% of total number of nodes, and are directly linked with the quality of aerodynamic simulations. In these regions, cells are highly stretched to capture the gradient in the direction normal to the wall. Therefore, anisotropic tetrahedral elements would be very difficult to be generated in complex geometries [22]. The most reliable method is to create semi-structured prismatic elements in this region and three dimensional tetrahedral cells to fill the rest of domain. To construct a mesh in this hybrid manner, the viscous mesh generation method must have good balance between robustness, quality and time cost, moreover good adaptability to the isotropic tetrahedral mesh generator.

2.2 Review of Boundary Layer Mesh Generation Techniques

2.2.1 Advancing Layer Methods

The most popular method for viscous mesh generation is the so called advancing layer method, in which wall nodes and cells are marched along the boundary normal direction to generate high-aspect ratio elements in a semi-structured manner close to the surface, and then the rest of the domain is filled with an isotropic unstructured grid. Early attempts of this method were proposed by Lohner [32] and Pirzadeh [33]. The main difference is that while Lohner inflates the whole boundary layer mesh at once, Pirzadeh in turn proceeds on a layer-by-layer basis. The advancing layer method avoids the difficulties associated with the creation of nodes in classical unstructured mesh generation methods for highly stretched cells, and allows the mesh to be formed in a hybrid manner[34]. The extension of this method to three-dimensional cases using prismatic elements is conceptually straight forward, although very involved from an implementation point of view. Mesh generation in three dimensional cases is described by Pirzadeh [2] using tetrahedral cells, and by Kallinderis et al. [35, 36] using prismatic cells.

The layer-by-layer approach ideally has an advantage of in robustness and quality of the generated viscous mesh, because of the validity, proximity and intersection checking are performed simultaneously with the layer creation. If the checking criteria are not met, the advancement of the viscous layer is terminated locally, providing a high degree of flexibility to the algorithm. All viscous layers are advanced until a termination criterion is met, such as a maximum element aspect ratio or number of layers. This algorithm, when combined with accurate normal calculation and smoothing techniques [33, 36, 3, 37] may creates a smooth transition between the last prismatic elements and the tetrahedrons generated by isotropic volumetric mesh generators, which is highly convenient for CFD solvers. However, due to the local nature of the termination criteria, the new boundary generated by the advancing layer algorithm, can be anisotropic and irregular in shape, which may result in extra difficulties for the volumetric meshing process.

Moreover, this method can be time-consuming if the number of total viscous cells is large, since an intersection checking criterion has to be conducted for every new face of the front.

The whole-layer-inflation approach can be more efficient than the previous approach since in this algorithm the viscous cells no longer need to be checked layer-by-layer and the generation process can be very fast [9]. However, as this algorithm tries to generate the same number of prismatic layers for the whole model, or at least for the same surface, it requires a set of additional operations to “fix” the invalid elements. For instance, this algorithm is often associated with a shrinking procedure to retain the validity of viscous elements in the region where intersection with other elements of the front appear. This is accomplished by progressively reducing the total thickness of the layer [38, 9]. This kind of thickness reduction usually leads to a steep transition between the prismatic and tetrahedral elements, and locally modifies the stretching ratio of all the layers in narrow or highly constrained regions. In addition, this algorithm may have problems for meshing very complex configurations in which degenerated or very small surfaces are hard to avoid.

Alternatively other approaches generate the anisotropic layered mesh upon the creation of the volumetric mesh [39, 40, 41, 42]. In this method the surface elements of the isotropic volumetric mesh are displaced away from the surface. At this point viscous layers can be inserted either at once, or on a layer-by-layer basis. The advantage of this method is that the validity of the elements can always be ensured since it departs from a valid volumetric mesh. However, a robust volumetric mesh deformation algorithm is the key to achieve high quality meshes, especially when dealing with very complex geometries. This approach is not the most preferred option for conventional hybrid mesh generation, but it is more popular when used in conjunction with Cartesian meshes since the volumetric mesh is always created first.

2.2.2 Surface Normal Calculation

One of the key factors in boundary meshing generation is the determination of the marching direction, which is approximately, the normal directions of the surface nodes. Connell and Braaten [43] and Athanasiadis and Deconick [8] computed the node normals directly from the

Figure 2.2: Deflection of node normal due to non-uniform surface mesh at a 3D convex corner

analytic surface model. This is certainly more reliable when the background mesh is very coarse. But this advantage may not appear in many of CFD calculations as fine background meshes are usually required, furthermore the geometrical entities may not be available at any moment, as access to the CAD geometry is required. Then the most common way to obtain the normals is, first use two edges, $\vec{a}$ and $\vec{b}$ of each triangle cell from discrete surface mesh, and then average all the normals of the triangles sharing that node. Given a set of cells surrounding node p , the unit normal of each cell, N_i , is calculated as,

Given c_i set of cells surrounding node p , the normalised normal of each cell is calculated N_i as,

$$N_i = \frac{\vec{a} \times \vec{b}}{\|\vec{a} \times \vec{b}\|} \quad (2.1)$$

where $\vec{a}$ and $\vec{b}$ are two edge vectors departing from a common node of the cell. The unitary node normal N_p is obtained as,

$$N_p = \frac{\sum N_i}{\|\sum N_i\|} \quad (2.2)$$

However, unlike in two-dimensional grids where the calculation of node normals is straight forward and unique, in three dimensional cases, simple averaging may not be able to provide an accurate normal calculation for all the cases. This problem is more critical in regions with high curvature variations or when the back ground mesh has a large disparity in elements size, such as convex/concave regions where mesh refinement is usually required. An example can

be found in Figure 2.2. It is shown that, when the surface mesh is not uniform, the calculated normal $\tilde{N}_p$ may deflect away from the correct one (N_p).

A general solution consists of considering different weighted averages of the surrounding cells to overcome this problem, see Frey and George [37],

$$N_p = \frac{\sum w_i N_i}{\|\sum w_i N_i\|} \quad (2.3)$$

where w_i is a weight factor which can be defined in various ways, for instance, (note that Equation 2.2 is a particular case of Equation 2.3)

- cell normal is weighted according to its area.
- cell normal is weighted according to the inverse of its area.

These variations may partially overcome the problem, but there is no guarantee the existence of a correct universally valid weighing method for all the cases. A notable finding to determine a valid normal, known as the visibility criterion, was proposed by Kallinderis and Wards [35]. Figure 2.3 illustrates the normal cone which is formed by the surrounding cell normals, and the visibility cone. It can be seen that the combination of different weights can result in the computed normal at a point p , inside of normal cone but outside of the visibility cone. Therefore, this normal is not valid for boundary mesh generation. It can be concluded that a valid node normal should be visible by all its surrounding cells, and the optimal node normal should be equally seen by all its surrounding cell normals, and obviously, this normal satisfies the visibility criteria. Pirzadeh [2] proposes an iterative method to find this optimal normal by looking for the normal that minimises the maximum angle for the given set of normals. Analogously as in the weighted method, different weights are assigned to each cell normal, depending on the angle created with the current computed node normal. If an angle is higher than the averaged angle of all surrounding cells, more weight is assigned to that cell. Therefore, the new computed normal tends to lean towards that cell normal to reduce the maximum angle. A simple sketch is illustrated in Figure 2.4 (a).

Alternatively, Aubry and Lohner [3] proposed a method with a higher insight of the complete

Figure 2.3: Sketch of the visibility cone and the normal cone at a 2D convex corner

Figure 2.4: Two optimal normal calculation methods based on the equal visibility criteria proposed by (a): Pirzadeh [2] and (b): Aubry and Lohner [3]

problem. The basic idea of this method is shown in Figure 2.4 (b). All the unit cell normals are lying on the same unit sphere. The problem of finding the normal that minimises the maximum angle for the given set of normals is cast into the problem of finding the point which minimises the maximum arc length on this sphere. Then, the optimal normal searching is turned into a famous geometrical searching problem, namely: finding the smallest circumscribed circle to the plane convex polygon created by the boundary of the convex hull of a set of points. The searching algorithm is given by Papadopoulos [44]. This method provides the most precise normal calculation, but also may be complex to implement than fore-discussed, since many geometric computations are involved.

2.2.3 Numerical Treatment of Surface Normals

In order to further improve the quality of generated mesh, the marching directions have to compromise with neighbour nodes especially at the regions where node normals have very acute variations in directions, such as sharp convex ridges and concave corners. Traditionally, these problems are solved using different forms of Laplacian smoothing (Equation 2.4), where the marching normal is smoothed by its neighbours. This scheme provides a gradual smoothing in the normal direction as the number of boundary layer mesh increases.

$$\tilde{N}_p = (1 - \beta)N_p + \beta \frac{\sum N_{p-neighbour}}{\|\sum N_{p-neighbour}\|} \quad (2.4)$$

Meanwhile, other methods are also developed to improve the quality of boundary viscous meshes in these regions. Multiple normal approaches are developed by Galimella and Shephard [38], Ito and Nakahashi[41, 45] and Aubury and Lohner [46]. The comparison with non-smoothing and the Laplacian smoothing (Equation 2.4), is illustrated in Figure 2.5. The basic idea of this method is to use multiple normals at convex ridges instead of a single one in order to obtain smoother normal variations. However, this approach may generate very tiny elements at convex corners, and special treatment should be done to avoid exposing quadratic cells to isotropic tetrahedral mesh generator (see Figure 2.5 (c)). Therefore, it does not show significant improvement in solution accuracy than a simple smoothing method in complex three-dimensional models.

The quality of boundary viscous mesh at concave corners can be enhanced by means of the directional enrichment method [47, 48] or the suppressed marching direction method [49, 24]. This method concentrates nodes near concave corners so that the viscous mesh in these regions can be generated in a hexahedral structured fashion. This kind of method for concave configurations is attractive but gives rise to a considerable increase in the mesh size without a significant impact on the results[8, 46]. Moreover, in very complex geometries with large number of ridges, this method may be hard to apply.

Although these kinds of hybrid viscous mesh generation techniques have been developed over

Figure 2.5: Different normal treatments at convex ridges: (a) non-smoothing; (b) with Laplacian smoothing; (c) multiple normal approach

Figure 2.6: Different normal treatments at concave corners: (a) non-smoothing, (b) with Laplacian smoothing, (c) direction enrichment

twenty years and have been successfully implemented into many commercial or open source codes, many improvements focused on a relatively narrow field of problems (e. g. seeking for optimal normal). More importantly, all these advancing layer kind algorithms heavily rely on the validity checking process which can be very sophisticated and time-consuming. Despite the many improvements and variations which have been made to improve the efficiency of the conventional methods, in general, the robustness and the efficiency has to be frequently traded. Therefore, if efficiency is favoured in the method design at the expense of of eluding the solution of complex geometric problems, which may be encountered for instance in complex internal cooling geometries of gas turbine blades, the method can be weak. In this case, more robust checking procedures are needed to ensure the validity of the generated elements, but wall-clock time may increase drastically.

Hereby, a very robust and efficient method to generate a viscous boundary layer mesh optimised for three-dimensional turbine internal cooling geometries is presented. Surface triangular cells

are used as input. From a surface mesh, the near wall viscous mesh is constructed in a prismatic manner based on an improved advancing layer method. The viscous meshes are generated in a layer-by-layer manner combining an improved validity checking algorithm to ensure the robustness for internal geometries, and a so-called pre-layer checking method in order to avoid extensive search and intersection routines intrinsic to all advancing-layer techniques.

A pre-processed checking procedure is conducted in a similar way as in the all-inflated approach, but its purpose is to identify the cells which will not show intersecting problems rather than generating a valid proper mesh. In this way, the validity and intersection checking methods are only applied to those cells which failed to pass the previous test, reducing the total number of checking and, therefore, computational cost. This hybrid mesh generation method has been tested in several representative geometries to demonstrate its capability and performance. The algorithm has proved to be much faster than the baseline method, while inheriting its robustness.

2.3 Surface Mesh Generation

Most grid generation techniques require the domain surface meshing prior to the generation of the volumetric mesh. Hence, surface grid generation is an essential and important step in the unstructured grid generation procedure. Surface meshes influence the volumetric mesh close to the boundary, and therefore they must have the same smoothness and continuity requirements as the volumetric mesh. But in addition, they are required to conform to the geometry surface and must accurately resolve high curvature regions.

The grid generation process begins by defining the geometry in Computer-Aided-Design (CAD) systems. Points are distributed on the edges that conform the surfaces. A surface grid is then generated on the boundary surface. These surface grids are generated in the parametric space using additional information to account for surface curvature. In the present work, the surface mesh is generated by means of a Steiner triangulation method described by Corral et al. [5]. The system uses an approximate physical space grid named MGG (Mapped space Generated Grid), following the terminology of Marcum [50]. The true surface definition is only used at the

Algoritmo 2.1 Surface mesh generation scheme by means of Steiner triangulations [5]

1. *Generate a grid entirely in mapped space using the standard planar two-dimensional Delaunay procedure. Obtain the physical space coordinates at the mapped space coordinates using a geometry definition routine. This defines the surface grid MGG generated entirely in mapped space.*
 2. *Obtain the physical space coordinates at the MGG coordinates using the CAD definition routine.*
 3. *Generate a first grid in physical space containing just the boundary nodes. Elements placed outside the domain are retained in order to ease search procedures.*
 4. *Triangles not satisfying the element size or curvature criteria are subdivided inserting new nodes in its mapped space centroids. Physical coordinates are obtained by linear interpolation in the MGG grid.*
 5. *A sequence of swaps is started in order to meet the min-max or max-min criteria. This local reconnection process is repeated until convergence.*
 6. *Smooth the node coordinates in physical space. Simultaneously the element size function is smoothed as well in order to obtain a better approximation of it.*
 7. *Critical node removal and high degree node elimination is performed.*
 8. *Steps 4-7 are repeated until convergence.*
 9. *Coordinates in the mapped space are transformed to physical space using the specified geometry definition.*
 10. *Elements outside of the computational domain are eliminated if required.*
-

beginning and the end of the process. In fact, the present viscous mesh generation process has no dependency on the surface method generation method as long as it is defined by a triangular mesh. The extension to surface meshes made up of quads is straight forward but it is not presented in this work. This basic steps in the overall procedure are listed in Algoritmo 2.1 and a surface mesh example obtained with this method can be found in Figure 2.7.

2.4 Viscous Mesh Generation

The boundary layer meshing approach described here departs from a standard advancing layer method, but incorporates an improved validity checking algorithm to cope with complex internal geometries. The generation procedure takes a surface triangular mesh as input, and grows a prismatic viscous mesh on it in a layer-by-layer basis [2] along the node normal direction.

Figure 2.7: Surface mesh of a leading edge impingement cooling channel with film cooling holes

Once the viscous grid is generated, the last front of the viscous mesh is passed to the volumetric mesh generator to fill the rest domain with isotropic tetrahedral elements. This procedure is opposed to the practise of generating all layers at once, and then checking/fixing invalid elements at the end [32, 9]. The layer-by-layer approach was found to be more robust since trying to generate all layers at the same time would encounter some difficulties because of complex geometries. Instead, the necessary quality and validity checking criteria are performed prior to each layer is actually generated, so errors do not propagate in the subsequent layers[8].

In present work, a viscous mesh generation method has been developed with the ultimate goal of meshing internal cooling geometries, many treatments were implemented to deal with this particular problem such as projections, validity checking and other problems due to the highly non-uniform surface mesh.

The main steps of the baseline viscous mesh generation method are described as follows, referring to Figure 2.8:

1. From an input surface mesh, viscous fronts and non-viscous entities are identified for the

Figure 2.8: Flow chart of viscous mesh generation

mesh generation process.

2. Calculate the thickness of each individual viscous layer.
3. Calculate the normal direction based on surface mesh to determine the growth direction. Special normal treatments are required to deal with the non-uniform surface mesh and the presence of non-viscous entities.
4. Since the viscous mesh is built in layer by layer basis, thickness correction and smoothing techniques are required to enhance the smoothness of each layer
5. Grow the layer and ensure its validity. Bad elements will be eliminated and pyramid or tetrahedron elements are used as transition to avoid the exposure of quadratic highly stretched elements to the isotropic tetrahedral mesh generator.
6. Use the last viscous layer as the front to create the volumetric mesh.

2.4.1 Face Identifications

The viscous mesh generation method follows an advancing layer approach to build prismatic cells in the near wall regions. The mesh, as it has already been mentioned, is constructed layer by layer in order to enhance the smoothness of the viscous mesh and the robustness of the process. Usually in the 3D computational domain there are always a few faces (for instance inlet and outlet patches) which behave as non-viscous, and multiple front issues may appear

Figure 2.9: Identification of faces and edges

if periodicity is considered in the model. Therefore, a process to identify in advance viscous fronts must be implemented. Besides, due to the presence of non-viscous faces, not all the faces and edges behave in the same manner during the viscous mesh generation process, extra surface and edge identifications are then also needed.

The front identification process is started from an unchecked viscous face, then a loop over edges to look for its neighbouring viscous faces is conducted. A viscous face which shares a mutual viscous edge with any viscous face in a front will be associated to the same front. This process is performed sequentially until all faces are checked. Then, the non-viscous faces and edges are identified in the following way, as illustrated in Figure 2.9

- Edges belonging to a viscous face and non-viscous face are identified as interface edges
- Edges which belong to two viscous faces are considered viscous edges
- Edge who belong to to two non-viscous faces are labelled as inviscid edges

These faces and edges require different treatments during the viscous mesh generation process. In this sense:

- A non-viscous face with all inviscid edges is considered an inviscid face and its surface mesh is conserved during the viscous meshing process
- A non-viscous face with at least one interface edge is an interface face. In this case, surface mesh has to be removed and re-meshed once the viscous mesh generation is completed.

Figure 2.10: Thickness of each viscous layer controlled by the growth factor β (left: $\beta > 1$, right: $\beta \approx 1$.)

2.4.2 Node Normal Spacing Specification

Viscous mesh generation demands very specific information to define the desired normal grid spacing in the region close to the viscous boundaries. Node spacing represents the way to concentrate viscous nodes near walls and can be specified in the following ways according to different priorities [51]:

1. **Control the first layer thickness:** The first layer thickness can be estimated using Reynolds number, hydraulic diameter and desired y^+ value in a classic CFD simulation. In this case, the first layer thickness, the total number of layers, and growth factor are specified. The thickness of other layers and the total thickness of viscous layers are then calculated accordingly.
2. **Control the total thickness of boundary layer mesh** in order to adjust the interior domain of the model. In this case, the total thickness of viscous mesh, the total number of layers and the growth factor are specified. The thickness of each individual layer is calculated accordingly.

A growth factor β is selected to be a positive number larger than one, and it is used to control the concentration of nodes. As shown in Figure 2.10, a large β indicates that the increment of thickness for each individual layer is small in progression. Therefore, the final viscous mesh is more uniform (lower case). In contrast, if β is very close to one (upper case), the thickness of each individual layer has larger growth rate by progression. In this case, nodes can be saved to reduce the size of the mesh and a smooth gradient from viscous mesh to isotropic mesh is acquired.

Algoritmo 2.2 Iterative weighted normal calculation

Compute the initial normal based on $N_p = \frac{\sum w_i N_i}{\|\sum w_i N_i\|}$, equal weight was used, $w_i = \frac{1}{n}$, where n is the total number of cells surrounding point p .

1. Evaluate the angles between the cell normal and node normal:

$$\alpha_i = \arccos(N_p \cdot N_i) \quad (2.5)$$

2. Calculate the averaged angle and the deviation $\theta_i = \alpha_i - \alpha_{avg}$.

3. The new weight is calculated based on the deviation from the averaged angle:

$$w_i^{new} = w_i^{old} \left(1 - \frac{\theta_i}{\alpha_{avg}}\right) \quad (2.6)$$

4. The new normal is calculated using the new weighted function, a relaxation factor β can be used:

$$N_p^{new} = (1 - \beta)N_p + \beta \frac{\sum w_i^{new} N_i}{\|\sum w_i^{new} N_i\|} \quad (2.7)$$

2.4.3 Determination of Marching Direction

The calculation of normal direction is an essential factor in the viscous mesh generation process, as it has direct impacts on the success of viscous layer growth. Considering the complexity and non-uniformity, simple evaluations may lead to the correct calculation of marching direction.

2.4.3.1 Weighted normal calculation

In the present method, the basic cell normal obtained by using the cross product of its two edges cells. And the node normal is evaluated using the weighted method proposed by Pirzadeh [2] with a slight modification, since its iterative manner is more suitable for coding.

The standard implementation is described in Algoritmo 2.2

The above scheme can be invoked for a few iterations till the averaged angle is smaller than a certain value. Its variation includes evaluating the new weighted function in a simpler way as:

$$w_i^{new} = w_i^{old} \left(\frac{\alpha_i}{\sum \alpha_i}\right) \quad (2.8)$$

Other techniques like multiple normals or directional enrichment were not used due to the reas-

Figure 2.11: Normal treatments at non-viscous edges

ons discussed in the review section, since it only has limited improvement in the solution but substantially introduce complexities to meshing system. Especially, normal smoothing is usually applied with number of grown layers, making the effort of extensive normal calculation reduce. In our experience, even with all the state-of-art methods in literature applied, there is still chance that the appearance of degenerated faces or bad parametric entities inevitably result in valid normal impossible to find. Instead, the flexibility in the element validity checking process to eliminate the invalid cells guarantees that the valid grids can be generated for any configurations.

2.4.3.2 Treatments for non-viscous entities

Due to the presence of non-viscous surfaces in internal cooling problem normals need special treatment on the edges of viscous and non-viscous contacted surfaces. Figure 2.11 shows the normal treatments for non-viscous edges. The identification of edges and faces is illustrated in Figure 2.9. The marching direction of a viscous node on an interface edge is obtained by projecting the normal on the non-viscous surface. It is done by searching for the nearest node on the surface. Similarly, If an inviscid edge contains one or two viscous vertex, the normal will be forced to project on the edge.

Figure 2.12: Comparison of layer thickness correction: (a) without correction; (b) with correction

2.4.3.3 Normal smoothing procedure

From the second layer, a Laplacian smooth procedure (Equation 2.4) is applied to improve the quality when generation is advanced. Smoothing of interior growth curves is done by re-orienting each growth curves to the average of its adjacent growth curves. Smoothing of the boundary growth curve is done by a modified procedure that accounts for their general shape. In this procedure, straight line approximations of the growth curve and its adjacent boundary growth curves are used for computing a smoothed direction and closest points searches done to locate the nodes of the growth curve onto the model boundary. Multiple iterations of smoothing are used over each entity and over all the entities. Although this smoothing distorts previously well calculated normals, it also corrects cross-over in many cases and evens out shape and size variations in the boundary layer mesh.

In some particular cases, like film cooling tubes, the end of the tube has large angle which lead to bad quality mesh at outlet. Therefore, the normals of non-viscous nodes will be assigned more weight in the smoothing process to its neighbour nodes and less affected by its neighbours.

2.4.3.4 Thickness correction

Once the marching direction is determined. A thickness correction should be done to calculate the actual marching distance according to its angle against the cell normal. The desired thickness is corrected in order to have a uniform shape, as calculated in Equation 2.9, where n is the

n^{th} layers, N_p and N_c denote the node normal and cell normal at point p

$$\delta_c^n = \frac{\delta^n}{(N_p \cdot N_c)} \quad (2.9)$$

Figure 2.12 shows the comparison of generated viscous meshes without and with correction. It can be seen that, without the correction, the viscous meshes are screwed in the high curvature region which is not preferred.

2.4.4 New Layer Generation

Once the marching direction and distance are determined. The positions of the new candidate nodes can be computed along the marching direction. However, before the new prismatic layer is actually generated, a series of tests are essentially made for each surface triangular cell before its corresponding prismatic elements is accepted in the new layer.

Unlike in the in two-dimensional cases where the layer generation proceeds in a sequential manner, and the success generation of each cell is independent. The generation of prismatic element in three-dimensional models depends not only on the validity of itself, but also relies on the topology of its neighbours. For instance, one of these examples is illustrated in Figure 2.13. If a neighbour cell is failed to grow in 2D cases, the exposed edges of the new cells are still able to create isotropic triangle cells. However, in 3D cases, the isotropic tetrahedral mesh does not allow such topology with exposed quadrilateral surfaces (Figure 2.13 (b)). Another issue is that 3D surface cells are stored in arbitrary order, therefore when a cell is about to be generated, the growth availability of its unvisited neighbour is unknown. The main difficulty is to provide this information in a way that a grid for each entity can be generated without knowing every detail of the other entities and to assure that it is going to be compatible with them. If this process is not done properly, invalid topology of elements may be created and further fixing scheme will require a rolling back process which will lead to a decomposition chain to existing meshes. Compared to intensively studies on the innovation of normal treatments, the algorithms of validity checking were scarcely discussed [38]. Because if it does not necessarily appear in

Figure 2.13: 2D (a) and 3D (b) boundary layer meshes with exposed surfaces

Algoritmo 2.3 Validity checking of prismatic elements

1. Check the proximity of three new candidate nodes

$$\min(d(p'_i, p'_j)) \geq d_{min}$$

2. Check the aspect ratio of prismatic element

$$\text{avg}\left(\frac{d(p'_i, p'_j)}{\delta_c^n}\right) \geq AR_{min}$$

where δ_c^n is the corrected thickness for the n^{th} layer at node p_i

3. Check the volume of the prismatic cell by subdividing into three tetrahedral elements
 4. If any tests above failed, the cell is inactive.
-

all cases and extra checking procedure could consume significantly more time, this problem could be eluded at different levels in existing methods.

In this work, a very robust algorithm for validity checking and layer growing is presented. An important feature of this method is that the checking and growing process is conducted in a consistent way, surface grid independent and, therefore no modifications are needed to carry back the already generated semi-structured layers grids.

The full checking process consists of three parts:

1. Cell validity & quality: In this step, a checking is performed to determine the validity of each new generated cell. Firstly, a test to examine the distance of the three new candidate nodes, and the aspect ratio of generated prism is made in order to avoid poor quality elements. Then, the volume of the prism is checked by subdividing it into three tetrahedral elements. The

Figure 2.14: Sketch of the minimum requirements of the neighboring cells to form a prism in the target cell

process is presented in Algorithm 2.3

2. Compatibility of neighbouring cells: The success for the generation of a valid prism also relies on the topological properties of the neighbouring cells. Figure 2.14 shows the topology of a prismatic element and its neighbours. Let us consider we want to create the prism on the top of the triangle, c_1 , which has two types of neighbouring triangles: a triangle, c_2 , sharing an edge with the target triangle, and a cell, c_3 , sharing a node with the target cell. The minimum requirement for the creation of a triangular prism on top of the triangle, is that, at least pyramid can be created for c_2 type triangles, and a tetrahedron can be created for c_3 type cells. Similarly to the prism case, the volume of the pyramid is checked by subdividing it into two tetrahedrons. In this way, regardless of any possible termination of nodes p_4 and p_5 , and if prisms can be created for the triangles c_2 and c_3 , no roll back process is needed, since the validity of these auxiliary non-prismatic cells is guaranteed. The implementation is illustrated in Algorithm 2.4.

3. Proximity/Intersection Checking: Last but not least, when layers are added to the front, proximity/intersection problems with other elements of the front inevitably appear. Figure 2.15 shows two typical intersection problems for non-neighbouring elements which can not be detected just by checking the prismatic element itself. This problem is very common for meshing in constrained spaces, where viscous layers coming from different surfaces or fronts are prone to clash. In our algorithm, the basic proximity-intersection checking method uses an alternating

Algorithm 2.4 Compatibility checking of neighbour elements

1. Get all c_2 type cells for c_1
 2. For every c_2 type neighbour cells:
 - if c_2 has already been built, skip test
 - if c_2 has not been built as prism, check the volume of pyramid $\text{vol}(p_2, p_3, p'_3, p'_2, p_5)$
 3. Get all c_3 type cells by finding cells who share only one node with c_1
 4. For every c_3 type neighbour cells:
 - if c_3 has already been built as prism, skip test
 - if c_2 has not been built, check the volume of tetrahedral $\text{vol}(p_2, p_4, p_5, p'_2)$
 5. If any tests above failed, the cell c_1 is inactive.
-

Figure 2.15: Intersection problems for neighbouring elements: (a) intersection within the same front, (b) intersection with a different front

digital tree (ADT) algorithm to minimise the number of candidates for checking. The details and implementation of the algorithm can be found in [52]. The algorithm is applied by storing all the surface viscous nodes in the ADT data structure. Each time a new node is considered to be included in the mesh, its location is tested against the ADT for proximity detection. If this new node is actually generated, it replaces the position of the previous node of its corresponding normal in the ADT. However, since the mesh of the front is large, disparity in the size and anisotropy of the elements are usually found, and therefore it is difficult to find out an universal definition for the size of the bounding box of the nodes. As a consequence, the ADT searching procedure can not be used as the sole criterion to detect proximity and/or intersection with other elements, and it is only used to provide a set of candidate nodes for checking. Additional tests will be made to examine the actual distance and intersection of these nodes.

Figure 2.16: Sketch of proximity and intersection checking procedure

Algoritmo 2.5 Proximity and intersection checking procedure

1. For node p_i of cell c
2. Create bounding box based on current thickness δ_c^n
3. Check proximity nodes via ADT
4. Check the actual distance between test node and candidate nodes provided by ADT.

$$\min(d(p_i, q_j)) \geq d_{min}$$

5. Check if the node p_i inside of the set of prismatic elements E_j , who contains q_j (neighbour cells are not considered)

- if E_j has already been built, check the sign distance to each surface of the prism. $\text{sign}(p, E_j)$. If the distance is too close, or all have the same sign, the test node deactivate.
- if E_j has not been built, create ghost elements $\tilde{E}_j$, check the sign distance to each surface of the prism. $\text{sign}(p, E_j)$. If the distance is too close, or all have the same sign, the test node deactivate.

6. Check if any edge has intersection with prismatic elements E_j .
 7. If any of above tests failed, the cell is inactive.
-

An example of the proximity and intersecting checking procedure is displaced in Figure 2.16. When a new node p'_1 is about to be generated, the ADT searching provides a candidate node q'_2 with the possible proximity problem. For the candidate nodes reported by ADT, the actual distance between the test node and the candidate node is checked first for quality considerations. Then, the intersection point is checked with the prismatic element (E_q) who contains the candidate node.

There are many standard ways to detect if a node is inside a volume, also known as the point-in-polygon (PIP) problem, for instance the ray casting algorithms [53, 54] or the winding number algorithm [55]. However, these algorithms are generally used to search nodes in arbitrary geometries, we only need here to deal with prismatic elements. Thus, the detection algorithm can be used in a simpler way by calculating the signed distance of the node to each side of the prism. If all the sides have the same sign, the node is inside of the prism. Another advantage of this simple algorithm is that, the distance to a surface can be also used as a factor to control the quality of the mesh. Although the node may be outside of prism, but it is too close to a surface, it will be detected and eliminated.

The last step is to check the intersection between the segments who contain node p'_1 against prism E_q . Similarly, on the contrary for the classical ray-triangle intersection algorithm [56], we use a simpler and faster algorithm by taking a few nodes on the segment and checking whether they are inside of the prim. This algorithm is simple but effective.

In addition, it is worthy to notice that, we did not use the well known triangle based ADT, which normally is used in classical advancing front methods. This is because, in the advancing front method, if a intersection problem happens, then the cell is not created. In this approach, because of that the activity of a node has direct impact to its neighbours, we also need to test other possibilities, like generating pyramidal and tetrahedral elements. This simple but efficient algorithm is proved to be faster than that used for classical triangle-triangle intersection problem. The implementation of the algorithm is presented in Algoritmo 2.5

2.4.5 Additional Treatments

2.4.5.1 Projection and building of transitional elements

Once the new nodes are created, a projection is required on the non-viscous entities. Similar to the process described in subsection 2.4.3.2, the projection is done by searching for the nearest node on the non-viscous surface or on the edges. In addition, in those cells which failed to generate prismatic elements transitional elements between different layers have to be built in order to avoid exposing the volumetric mesh generator to highly stretched quadrilateral faces, as advocated by Connell and Braaten [43]. If a cell has two active nodes, a pyramid element is built. If a cell has only one active node, a tetrahedral element is built. Only one transitional cell will be built and this cell will be de-activated for the next layer.

2.4.5.2 Layer generation for periodic cells

The presence of periodic surfaces is a significant characteristic in turbomachinery applications. The mesh in two periodic surfaces is not necessarily the same unless specific measures are taken in the whole mesh generation process.

Periodic surfaces are generally treated as non-viscous entities. To illustrate the procedure, a simple sketch can be seen in Figure 2.17. Pairs of periodic nodes are coloured in red and blue. In addition to the tests described above, the generation of periodic patches must be explicitly taken into consideration since the generation of periodic cells on each side should be synchronised. No master or slave relations are used here, when a cell pass all tests, the generation of its paired cell will also be tested, and if either of the nodes of the paired cell fails for some reasons, both cells will be set as de-activated.

In addition, there is a potential problem which is usually overlooked, namely the ordering problem. The fact that cells in unstructured grids are stored in an arbitrary order, may eventually provoke the algorithm failure. For instance, the cells on the viscous faces are stored and visited in the order described in Figure 2.17. If the non-periodic cells c_1 and c_2 are generated first. Then, considering the topology with their neighbours, c_4 will be generated, since all its three

Figure 2.17: Sketch of periodic faces

nodes are active. However, if c_3 fails to grow, for some reason, the deactivation of c_4 will inevitably lead to a roll back process for c_1 and c_2 . A simple way to solve this problem is to store all periodic cells and force the scheme to check the periodic cells prior to others.

Finally, the generation scheme is proceeding layer by layer, until reaching the target number of layers or all cells are deactivated.

2.5 Accelerating Method

In general, all advancing layer meshing algorithms heavily rely on the intersection checking process, which can be sophisticated and time-consuming. In conventional advancing layer methods, a trade-off between robustness and efficiency is usually performed. In the robust viscous generation method described above, despite the systematic usage of ADTs substantially reduces the number of operations in the proximity/intersection procedure, this process is still responsible for most of the time required to obtain a high-quality viscous mesh. Therefore, the key element of an efficient advancing-layer method is to reduce the total number of calls to this method while maintaining the same level of robustness.

In this section, a so called Outer Layer Boundary (OLB) checking routine is proposed in order

Figure 2.18: Pre-layer checking, illustrated in 2D

to reduce the number of element-to-element intersection checking intrinsic to the advancing-layer techniques. It is concluded that while the cost of standard advancing layer algorithms is $O(N_s \times N_L)$, where N_s is the number of surface elements and N_L the number of layers, the asymptotic cost of the new “Skyline” method is $O(N_s)$ for large enough number of layers, which is significantly more efficient than the baseline method.

The Outer Layer Boundary checking procedure is conducted in a similar way as the inflated approach [57], but its purpose is to identify the cells which will not pose intersection problems rather than generating a proper valid mesh. Then, the checking process has only to be applied to those cells for which the OLB checking fails, drastically reducing the total number of checking and, therefore, the computational cost. This hybrid mesh generation method has been finally tested in several representative geometries to demonstrate its capability and performance. The new algorithm has proved to be much faster than the standard advancing-layer method for an equivalent robustness.

2.5.1 The OLB Checking Algorithm

Experience shows that even in the most complex models, there are many viscous cells which are able to grow until the outer boundary of the generation process without any interference

Figure 2.19: Illustration of the outcome of the outer layer boundary checking pre-processing step for a ribbed cooling passage. Red: cells which passed the test. Cyan: cells which failed to pass the test.

with other elements of the front. This effectively means that the computationally intensive proximity checking process could have been avoided for these cells. Algorithm efficiency can be substantially improved if these cells were removed from the very beginning from the proximity/intersection checking procedure. According to this observation, a novel advancing-layer method is proposed. The proposed algorithm is based on the idea of identifying beforehand the surface cells that will not need to be checked to reduce the computational time. This identification process is conducted in a pre-processing step by the “Outer Layer Boundary Checking” algorithm.

Figure 2.18 illustrates this “Skyline” scheme in a 2D configuration. The OLB checking scheme uses the standard layer generation and checking algorithms to calculate the virtual prismatic cells whose base is the surface element and its height the total boundary layer thickness set by the user. Aspect ratio and quality control are deactivated so that high aspect ratio elements are allowed in this phase. As shown in Figure 2.18, the generation of valid points (coloured black) is guaranteed regardless of the other cells. Therefore, the proximity/intersection checking for those elements is no longer needed. Only those points, which failed to pass the OLB layer checking, for instance the cells located at the corners and small gaps, are considered for a full intersection checking in the standard grid generation procedure.

Algoritmo 2.6 OLB checking algorithm

-
- **Step 0:** ADT is initialised with all the cells from original surface triangular mesh.
 - **Step 1:** Generation of a unique virtual viscous layer based on the total thickness of the prismatic region. Normal vectors are not smoothed to obtain a conservative estimate.
 - **Step 2:** Loop over all viscous surface cells to perform the following Proximity/Intersection checking. For each candidate cell returned by the ADT, check the actual distance and intersection with the inflated boundary. If any node of this front is too close to any other node of the estimated outer boundary, or the corresponding normal vector intersects to any of these ADT cells, this prismatic cell will be considered invalid.
 - **Step 3:** For each cell which failed to pass the test, the distance between this cell and its problematic neighbour is stored in order to approximate the index of layer in which proximity/intersection will occur when the actual viscous mesh is generated ($n_{problem}$).
 - **Step 4:** Store the list of cells which did not pass the test.
-

This OLB checking reduces computational time in two ways. Firstly, the proximity/intersection detection algorithm is only performed for the cells which fail to pass the OLB test. In general this is only a small fraction of the total number of viscous cells. Secondly, the ADT-searching algorithm contains only cells which fail to pass the OLB algorithm, which are much less than the total number of cells and, therefore, the ADT is lighter, and the time required to update the ADT after each new insertion is significantly reduced because contains fewer data.

Figure 2.19 illustrates the outcome of the pre-processing algorithm for a two-pass ribbed cooling channel. Red coloured cells are the cells which passed the OLB checking. It can be seen that the number of red cells is much larger than that of the blue cells, which are the ones selected for a full checking. It can be observed that these cells are concentrated in the edges and corners where there are steep curvature changes.

The overall description of the OLB checking algorithm is listed in Algoritmo 2.6.

After performing the OLB checking, the original set of surface cells is classified in two groups: (i) the valid cells, where the layer growth will only be stopped by mesh quality and neighbour validity criteria; (ii) the cells which have failed the OLB checking test, and that will undergo a full proximity/intersection checking during the actual mesh generation process. The proximity/intersection process is done in the same way as described in the previous section. The only difference is that the ADT is in this case only built with those cells which failed the OLB check-

Algoritmo 2.7 Layer generation process of OLB method

- **Step 0:** ADT is initialised with cells which failed the OLB checking.

while $N_{\text{active-nodes}} > 0$ or $n_{\text{layers}} \leq n_{\text{target}}$

- **Step 1:** A normal vector to the surface is associated for each boundary node. Normals are smoothed when $n_{\text{layers}} > 1$.
 - **Step 2:** Create a new virtual prismatic cell for all viscous surface cells by inserting new nodes along each normal with a desired spacing.
 - **Step 3:** If cell passed OLB pre-processing step, perform following checking,
 - Cell validity & quality
 - Neighbour validity
 - **Step 4:** If cell failed OLB checking, perform following checking,
 - If $n_{\text{current layer}} < n_{\text{problem}}$
 - * Cell Validity & quality
 - * Neighbour validity
 - If $n_{\text{current layer}} \geq n_{\text{problem}}$
 - * Cell Validity & quality
 - * Neighbour validity
 - * Proximity/Intersection
 - **Step 5:** If cell passes all generation checking $\implies$ generate new prismatic cell,
 - If cell passed OLB checking
 - * Update mesh structure
 - If cell failed OLB checking
 - * Update mesh structure
 - * Update ADT
 - **Step 6:** If cell fails generation checking $\implies$ deactivate nodes, generate prism/tetrahedral to maintain consistency.
-

ing. As it has been previously explained, by doing this pre-processing the computational time can be substantially reduced since the total number of cells undergoing intersection checking algorithms is lower, therefore, the number of ADT updates, which is the most expensive part of this process, is consequently decreased as well.

If a cell passes all the tests, a new prismatic element is built, otherwise according to the number of the active nodes in a cell either a transitional pyramid or tetrahedron is constructed to avoid leaving a quadrilateral or a bad quality triangle surface cell exposed to the volumetric mesh generator. Independently of whether a cell passes or not the OLB pre-checking test, all the cells are created on a layer-by-layer basis. The decision of a cell remaining active or not is taken in a local manner to ensure that in any case the topology of the final mesh is such that the layer can be completed safely (see Figure 2.14). The layer generation and validity checking process is illustrated in Algoritmo 2.7.

2.5.2 Algorithm Complexity Analysis

The objective of this subsection is to understand the origin of the efficiency of the proposed approach. The computational cost of the advancing-layer method is proportional to the number of calls to the proximity/intersection detection procedure, which is the most expensive part of the algorithm,

$$Cost_{baseline} \propto (N_s \times N_L) \log(N_s) t_{cell-check} \quad (2.10)$$

where N_s is the number of elements of the surface grid, N_L the number of viscous layers and $t_{cell-check}$ the time required to make the intersection checking for a single cell. The previous expression assumes that an ADT is used to assist this process, and that the cost required to traverse the ADT scales with the logarithm of the total number of points.

The computational cost of the proposed method can be split in two parts: the first is the so-called Outer Layer Boundary (OLB) pre-processing step, which tests all the surface elements and hence its cost is $Cost_{OBL} \propto N_s \log(N_s)$. The second part is the actual full intersection checking, that is performed only for a fraction, α , of the total number of cells. This factor α represents

the ratio of surface cells that have failed the OLB pre-processing and will be fully checked. The value of α will depend on the geometry of the model, the quality of the surface mesh and the boundary layer thickness set by the user. The cost of the “Skyline” method can then be written as,

$$Cost_{Skyline} \propto N_s \log(N_s) t_{cell-check-OLB} + \alpha N_s \times (N_L - n_{problem}) \log(\alpha N_s) t_{cell-check} \quad (2.11)$$

where the effect of different efficiency of the ADT and the different time required to make the intersection checking for a single cell, associated to the different sizes during the first and second part of the algorithm have been retained. The relative cost between both methods is then,

$$\frac{Cost_{Skyline}}{Cost_{baseline}} \propto \frac{t_{cell-check-OLB}}{N_L t_{cell-check}} + \frac{\alpha (N_L - n_{problem})}{N_L} \quad (2.12)$$

For the OLB pre-processing, each ADT-node is associated to a larger bounding box than the typical one employed during the standard advancing-layer method method. This is done in order to maintain robustness even for those cases in which large differences in size between the surface mesh and the boundary layer thickness appear. Due to this, the proximity/intersection algorithm during OLB method has to deal with a higher number of candidate nodes returned by ADT and, therefore, time required to make the intersection checking for a single cell increases. Empirically, it can be seen that $t_{cell-check-OLB}$ and $t_{cell-check}$ are related by a factor between 2 and 3. A limit estimation with α equal to 0, i.e., a case in which all the cells have passed the OLB checking, “Skyline” method will be faster than the standard one by a factor determined by the number of total layers (Equation 2.13). For the opposite case, α equal to 1, “Skyline” method will entail a penalty due to the extra-cost of performing OLB pre-processing compared to the standard one (Equation 2.14).

$$\frac{Cost_{Skyline}}{Cost_{baseline}} \propto \frac{2 \sim 3}{N_L} \quad \text{for } \alpha = 0, \text{ and} \quad (2.13)$$

$$\frac{Cost_{Skyline}}{Cost_{baseline}} \propto \frac{3 + N_L}{N_L} \quad \text{for } \alpha = 1 \quad (2.14)$$

The efficiency of proposed method will be, therefore, a function of α and N_L . For a specific case, *ceteris paribus*, the higher the number of layers are generated the higher the speed-up factor achieved.

2.6 Volumetric Mesh Generation

Once the viscous mesh is generated, the last layer of the viscous mesh becomes a second surface grid made up solely of triangular cells. This new surface grid is used as the departing point to generate an isotropic volumetric mesh to fill the rest of the domain. This tetrahedral volumetric mesh is generated by means of a three-dimensional Delaunay algorithm. The final mesh consists of prisms, transitional pyramids and/or tetrahedrons, and isotropic tetrahedrons.

2.7 Test Cases

Different meshing examples are presented in this section to illustrate the efficiency and effectiveness of the proposed method. Complex 3D models were selected considering realistic industrial relevance, such as complex geometries derived from “dirty” CAD models. All these tests have been conducted in a single core of an Intel Xeon E5-1620 CPU.

The first case presented here will be used to evaluate the benefits in terms of computing time achieved by doing the OLB checking procedure compared to the *baseline* algorithm. For our purposes, the standard advancing-layer method in which all the growing nodes will have to be checked will be referred to as the *baseline* algorithm.

2.7.1 Turbine Leading Edge Impingement Cooling Mock-up

The first case is an experimental mock-up of a leading edge impingement cooling channel of a turbine blade. This case has been selected because its inner domain presents large differences in size. The total thickness of the boundary layer mesh was intentionally set larger than the

Figure 2.20: Model and final mesh of impingement cooling channel

diameter of the impingement and film cooling tubes, in order to test the robustness of proximity/intersection algorithm. The initial surface mesh is composed of 775k triangles, and in total 20 viscous layers are generated. Figure 2.20 and Figure 2.21 depict the final mesh of the impingement cooling channel model. It can be seen that the algorithm detected the collisions in tube regions successfully, and a high quality mesh with smooth transition between the last prismatic cell and tetrahedral element is achieved. In total 11.6M viscous cells were created in 5.0 mins, which is quite remarkable.

An additional mesh was created to compare the mesh quality obtained with two different boundary layer thicknesses. This comparison is displayed in Figure 2.22 and the mesh statistics are shown in Table 2.1. As expected, the thickness reduction in the narrow regions allows to generate more viscous cells in less time than the previous case. This is because the reduction of the layer thickness significantly reduces the number of intersecting cells in the front. However, it is hard to obtain a smooth transition between the prismatic mesh and the volumetric tetrahedral grid.

Table 2.2 compares the execution time of the new and the *baseline* method. It can be seen that for *baseline* method the intersection checking and ADT regeneration consume most of the time. The speed-up factor of the new algorithm is between 4 and 8 with respect to the baseline

Figure 2.21: Plane-cut of the final mesh for the impingement cooling channel.

method. The acceleration of the method is obtained by reducing the number of intersection checking and the ADT updating time when new elements are added or removed from the front, which compensates the penalty incurred by the pre-processing step of the OLB scheme.

For this case, a representation of the computational cost of “Skyline” method compared to *baseline* is depicted according to Equation 2.12, assuming a factor of 2.5 between the $t_{cell-check-OLB}$ and the $t_{cell-check}$ and an averaged $n_{problem}$ of 5 (Figure 2.23). Two sets of tests were performed

Figure 2.22: Quality comparison. Mesh A: constant thickness with smooth transition (left), Mesh B: reduced thickness height (15%) at the narrow regions (right).

Impg. Model	N_s (N_L)	OLB success rate (time)	Visc. cells (time)	Total cells (time)
Mesh A	775k (20)	89% (2.7min)	11.6M (5.0min)	20.5M (12.4min)
Mesh B	775k (20)	98% (<1min)	15.1M (2.3min)	24.4M (5.5min)

Table 2.1: Grid size and wall-clock generation time of the impingement cooling channel.

Impg. Model (Mesh A & Mesh B)	<i>Baseline</i>	OLB—Checking	<i>Baseline</i>	OLB—Checking
Basic Common Algorithm (% Total Time)	6%	6%	6.7%	6.7%
Intersection Checking & ADT Regeneration (% Total Time)	94%	5%	93.3%	< 1%
OLB Checking (% Total Time)	N/A	14%	N/A	5.4%
Total Time	100% (19.6min)	25% (5min)	100% (19.0min)	12.1% (2.3min)
OLB success rate (% Total Surf. Cells)	N/A	89%	N/A	98%

Table 2.2: Comparison of the execution time between the *baseline* and proposed algorithm for the meshes A and B of the impingement cooling configuration.

with a different number of layers. For each set, α was modified by changing viscous mesh parameters in order to achieve a good sampling of the curve, keeping the same surface mesh. For both cases, it can be seen that data fit properly the cost curve modelled before and, as it was remarked, the efficiency of the proposed method compared to *baseline* algorithm increases with N_L .

2.7.2 NASA Common Research Model

The NASA Common Research Model (CRM) [58] has been commonly used as a standard test case to measure the relative efficiency of 3D grid generation tools. It is presented as the first meshing example despite our code was specialised for turbomachinery problems. Figure 2.24 displays the model surface and a cobbled plane around the CRM model, whereas Table 2.3 benchmarks the new method against two well-established grid generation tools. The *TriTet* [59] hybrid mesh generator builds the prismatic grid region in a standard layer-by-layer basis, whereas the rest of the domain is filled with isotropic tetrahedrons by means of a serial advan-

Figure 2.23: Accelerating factor comparison between “Skyline” and *baseline*. $\alpha = (1 - \text{OLB success rate})$.

cing front method. The commercial software *ICEM – CFD* contains multiple algorithms for hybrid mesh generation; in this case the selected algorithm inflates the entire prismatic region at once, then they split the created column into prisms. Finally the rest of domain is filled by means of a Delaunay tetrahedralization. Therefore, the two reference tools represent two classical viscous mesh generation algorithms summarised in the introduction section. The data presented here are taken from the work of Tomac and Eller [9]. For the present approach, two meshes with different viscous settings (growth ratio, thickness, etc.) were created to test the robustness and efficiency of the method under different conditions. In particular, the boundary layer thickness of the mesh B is the half of that of the mesh A. All the four meshes are departing from a similar surface mesh containing around 1.2 million triangles, and have 35 viscous layers.

It can be seen in Table 2.3 that, due to the different thickness of the prismatic grid layer, the percentage of viscous cells which pass the pre-processing OLB is slightly different, resulting in a different generation time and number of prismatic cells. However, in both cases the generation time of the present method is at least ten times less than that of the two referred algorithms. This is because, most of the cells ($>97\%$) pass the OLB checking step, and do not need to experience the proximity/intersection checking algorithm in the generation process, which is the most expensive part of the method. In addition, it can also be observed that a lower passing rate of the OLB results in a higher computational time, what proves the rationale of the proposed algorithm.

Figure 2.24: Mesh example of Common Research Model (CRM)

CRM	N_s (N_L)	OLB success rate (time)	Visc. cells (time)	Total cells (time)
<i>TriTet</i>	1212k (35)	N/A	34.3M (1h32min)	50.1M (7h20min)
<i>ICEM – CFD</i>	1212k (35)	N/A	42.4M (1h18min)	57.9M (1h51min)
Mesh A	1231k (35)	97% (3.4min)	36M (5.7min)	46.1M (8.7min)
Mesh B	1231k (35)	99% (2.47min)	40.5M (4.9min)	50.2M (7.8min)

Table 2.3: Grid size and wall-clock generation time of three different mesh generation tools for the CRM model.

2.7.3 Internal Cooling Channels of Turbine Blade

In an engineering design or analysis loop, the mesh generation process begins by importing realistic three-dimensional CAD models. The geometry exported from a CAD system is in most of the cases “dirty” due to numerical tolerance issues within the CAD system (often exacerbated by greatly differing scales within the geometry), format translators, and lack of numerical compatibility between different representations of the same object [7]. Such models consist of hundreds or thousands of complex warped faces, with a large size disparity, with degenerated or badly parametrised entities. These complexities in the grid generation process can make the generation of the mesh infeasible. These models can always be cleaned or simplified to make them compatible with the mesh generation process. However it requires a non-negligible amount of labour time, especially in an optimisation process, in which large number of models need to be processed. This test case represents one of these uncleaned real geometries.

Figure 2.25 and Figure 2.26 depict the leading edge, mid-chord, and trailing edge internal cool-

Mesh & no. face entities.	sur. cells & no. visc. layers	pre-layer passing rate & time	no.visc. prisms & gen. time	no. total cells & time
LE channel (1332)	418k (22)	28.94% (1.8min)	5.9M (4.2min)	10.6M (5.2min)
MC channel (452)	591k (22)	48.53% (2.0min)	10.4M (5.0min)	22.8M (8.0min)
TE channel (389)	350k (22)	44.14% (1.1min)	6.2M (4.4min)	10.5M (5.3min)

Table 2.4: Grid size and wall-clock generation time of three internal cooling channels of rotor blade

Figure 2.25: Surface mesh of three internal cooling channels of rotor blade

Figure 2.26: Mesh cut-off of three internal cooling channels of rotor blade

Figure 2.27: Temperature field of internal cooling channels of a turbine blade

Figure 2.28: Overall heat transfer comparison for leading edge cooling channel: CFD vs EXP

ing channels of a turbine rotor blade. These three channels come from realistic CAD models without cleaning or simplification. Therefore, they are composed of hundreds of face entities including many micro-surfaces. The plane-cut view of the final mesh is shown in Figure 2.26 and the mesh statistics are presented in Table 2.4. In accordance with our experience the present method is robust enough to be used in a production environment.

Finally some CFD simulations performed for this test to demonstrate that the whole analysis process can be conducted, and that the grid quality is high enough to be handled by an unstructured viscous fluid solver. Figure 2.27 depicts the temperature contour of the internal flow field of the three cooling channels, whereas Figure 2.28 illustrates the overall heat transfer per-

formance on the pressure side and suction side of the leading edge channel at various operating points. As presented, the numerical results provide in overall satisfactory results of the heat transfer trend against the experimental data.

2.8 Summary

This chapter describes the development and implementation of an efficient and robust advancing layer viscous mesh generation method for 3D complex configurations in an hybrid fashion. The final mesh is created in a prismatic-tetrahedral hybrid manner for computational fluid dynamics (CFD) calculations. We first give a brief introduction and review of the boundary layer mesh generation methods. Then implementation of boundary mesh generation method with enhanced validity checking is described in detail. Finally, a new approach is proposed to improve the efficiency of the baseline algorithm.

The accelerated method employs a pre-processing algorithm, called the Outer Layer Boundary (OLB) checking scheme, to identify in advance the cells which can be generated without performing expensive intersection checking methods against other cells of the front. The method substantially reduces the number of intersection checking operations compared to classical advancing layer methods, significantly reducing the mesh generation wall-clock time while maintaining the same level of robustness and quality than the baseline algorithm. For the cases presented here, it is concluded that the new method, named as the “Skyline” method, is between 4 and 10 times faster than the standard one. Moreover it is concluded that the new method is significantly faster as well than other well established hybrid mesh generation systems. Meshes for three different cases of industrial complexity have been presented. This approach has proved to be useful for large size 3D complex configurations where mesh generation time and interactivity are crucial.

3 Coupled Fluid-Solid Heat Transfer

Method

Contents

3.1	Introduction	58
3.2	Baseline Loosely Coupled Strategy	67
3.3	Analysis of Loosely Coupled Approach with Dirichlet-Robin Interface . .	70
3.4	Practical Case Study	83
3.5	Results and Discussion	86
3.6	Summary	93

This chapter describes a fast and robust loosely coupled fluid-solid heat transfer method for steady state heat transfer analysis. The method is based on the improvement of a loosely Dirichlet-Robin coupling approach. The analysis of the loosely coupled process using a one-dimensional model shows that the convergence rate is influenced by both, the physical and virtual Biot numbers. The impact of using incompletely converged CFD solutions has also been further investigated. The proposed has been validated against the analytical solution of a 2D coupled flat plate, and further applied to real turbomachinery configurations. The results have shown that the computational time has been substantially reduced compared to the baseline method, and it only spends around 35% more than a steady state CFD simulation, which is attractive for industrial applications

Figure 3.1: Development of high pressure turbine material versus turbine inlet temperature [4]

3.1 Introduction

3.1.1 Background

Gas turbines tend to use very high turbine inlet temperatures as a mechanism to increase cycle thermal efficiency and hence the overall performance of the engine. As the turbine inlet temperature increases, the heat transferred to the turbine blade also increases. Figure 3.1 depicts the development of high pressure turbine material versus the increment of gas turbine inlet temperature (TIT). Nowadays, the turbine entry temperature has reached more than 600K above the limit of the newest single crystal casting blade. The level and variation in such a high temperature level has a direct impact on turbine material, which ultimately undergoes thermal stresses and blade failures, and must be limited to ensure durability and achieve reasonable long-life operation. Therefore, additional to the development of advanced cooling systems, the accuracy of temperature prediction is also of paramount importance for turbine analysis. As the blade life maybe reduced by half if the blade metal temperature prediction is off by only 30K [60].

Conventionally, CFD and solid thermal conduction analyses have been performed separately at different stages of the design loop. The fluid solver provides a simplified estimate of the thermal

Figure 3.2: Schematics of the heat flux as a function of wall temperature when the h is of interest: (a) linear function, h is independence of T_w (b) nonlinear function, h is variant with T_w

characteristics at the wall interfaces, usually under the form of a heat transfer coefficient. Then this information is applied to solid thermal codes for metal temperature calculation, which in turn is further used to predict thermal stresses and displacements. Several hypotheses are implicit in this process and the accuracy of this kind of methods inevitably relies on empirical database and engineering judgement. The basis assumption is that the convective heat transfer coefficient is predominantly determined by the flow field and not significantly affected by the imposed boundary heating conditions[61]. Thus, the convective heat transfer coefficient h , is presented by the Newton's law,

$$q = h(T_w - T_{ref}), \quad (3.1)$$

where q is the heat flux, T_w is the wall temperature, T_{ref} is the reference temperature and normally, it takes the adiabatic wall temperature or flow bulk temperature. If h is only determined by the flow field and is independent of T_w , it implies that the wall heat flux varies linearly with the wall temperature, as shown in Figure 3.2 (a). It is worth noting that this scenario holds only if the energy equation and its boundary conditions are linear with temperature, and the continuity and momentum equations can be decoupled from the energy equation, as follows:

If the the Mach number is small, $Ma \ll 1$, the temperature differences are small, $(T_w - T_\infty)/T_\infty \ll 1$, and the body force is negligible, under these conditions the density is nearly constant, $\rho \simeq \rho_\infty$, and the governing equations reduce an incompressible fluid. One of the main consequences is

that the continuity and momentum equations decouple from the energy equation. The steady state continuity and momentum equation for an ideal gas can be written in a compact form as,

$$\begin{aligned}\nabla \cdot \mathbf{V} &= 0 \\ \rho_\infty \mathbf{V} \cdot \nabla \mathbf{V} &= -\nabla(p - p_\infty) + \nabla \cdot \bar{\bar{\tau}}\end{aligned}\tag{3.2}$$

The steady equation for the internal energy reads,

$$\rho_\infty C_v \mathbf{V} \cdot \nabla T = -p_\infty \nabla \cdot \mathbf{V} + k_f \nabla^2 T,\tag{3.3}$$

if the viscous dissipation is much smaller than the conduction, which is true if the Eckert number is much smaller than one, $Ec \ll 1$. The term $p_\infty \nabla \cdot \mathbf{V}$ cannot be neglected since by taking into account the small density variations, the continuity equation becomes,

$$\nabla \cdot \mathbf{V} = -\frac{1}{\rho_\infty} \mathbf{V} \cdot \nabla \rho,$$

which together with $\rho_\infty T_\infty = \rho T$, leads to $\rho_\infty \nabla T + T_\infty \nabla \rho = 0$ and therefore the energy equation becomes,

$$\rho_\infty \mathbf{V} \cdot \nabla (T - T_\infty) = \frac{k_f}{C_p} \nabla^2 (T - T_\infty),\tag{3.4}$$

where T_∞ is the incoming inlet temperature, the above equation has to be solved together with the boundary conditions, $(T - T_\infty) = 0$ where $\mathbf{x} \rightarrow \infty$, whereas at the wall $\mathbf{x} = \mathbf{x}_w$ $(T - T_\infty) = (T_w - T_\infty)$. The previous equation is linear in the temperature, and is readily shown that the heat flux is proportional to the temperature difference between the flow and the incoming flow. Let us summarise the hypothesis we have made to arrive to this conclusion:

1. The flow is quasi-steady
2. Mach number is small, $Ma \ll 1$, to neglect compressibility effects.
3. Temperature difference between the wall and the incoming flow is small, to ensure that the density is constant in first approximation, as well as the fluid properties such as C_p and k .

4. Ec number is sufficiently small, so that the viscous dissipation can be neglected

However, these conditions are hardly all-satisfied in highly complex flow cases with large temperature differences, such as HPT external flows, cavity flows or some turbine internal cooling problems. In such cases, the heat flux, may become a non-linear function of T_w (Figure 3.2 (b)), thus, traditional methods may lead to considerable errors in the estimate of the temperature field, and to severe logistic problems to handle the exchange of information among different codes. Many open literature can be found to demonstrate how heat transfer is influenced by the wall temperature distribution and the thermal property of the solids [62, 63, 64, 65, 66, 67]

In the problems which are concerned with radiation needs to be retained and therefore boundary conditions for the solid are non-linear, temperature differences between the solid parts and the coolant flow are of the order of themselves, since components are heavily cooled, viscous dissipation cannot be neglected for example in rotating cavities, and finally temperature distributions are not known beforehand and exhibit steep variations, further complicating the use of uncoupled methods. For all these regions the use of coupled methods is highly recommended, but there are a number of practical reasons which prevent, even today, the general adoption of this kind of methods.

3.1.2 Coupled heat transfer methods

Greater accuracy of temperature prediction can be offered by means of coupled fluid-solid heat transfer methods, in which fluid and solid heat transfer problems are solved simultaneously without the use of further hypotheses, since the heat flux and temperature continuity across fluid-metal interface is ensured by construction. Because of their relevance, coupled heat transfer algorithms have received considerable attention from different engineering perspectives in the last decades.

In some attempts, coupled codes were developed as extended CFD codes where the solid domain was included as a part of the whole computational domain. The governing equations are solved simultaneously in both domains, by explicitly setting to zero all the velocity compon-

ents, and using solid material specific heat coefficient and thermal conductivity in the solid domain. Some implementations and application examples can be found in [68, 69, 70]. This kind of monolithic approaches are numerically consistent and generally lead to stable solutions. However, as a particularisation of a CFD code, its efficiency as a purely heat transfer solver is limited, and relies on the algorithms of the original CFD solver, which usually are not designed to cope with the specific problems related with heat conduction.

A second class of methods, known as partitioned methods [14], employ specialised numerical methods to solve heat conduction in a more efficient way. In this kind of approaches, two separate codes, different algorithms may be used, are coupled at the interface while the fluid and solid domains are solved independently. In each coupling iteration, boundary information is exchanged between the two solvers. Upon convergence, the continuity of temperature and heat flux at the interface is ensured [13]. Depending on how the exchange of information at the common interface is treated, there are different approaches of partitioned methods: Dirichlet-Neumann interface, effectively meaning that the temperature is imposed at the fluid walls while heat flux is imposed at the solid side; a Neumann-Dirichlet interface is just the opposite [71]. Convective boundary conditions were also considered as well, leading to Dirichlet-Robin and Neumann-Robin type interface. a review of the different approaches can be found in [72]. Theoretically, the Robin boundary condition can also been applied to the fluid solver [73], however it has not been widely used since it is not a nature boundary condition for fluid solver. As partitioned methods require the exchange of information for two different numerical scheme, particular attention needs to be paid in this kind of approaches to avoid numerical stability problems.

3.1.2.1 Time scale disparity problem

A very important issue in coupled fluid-solid heat transfer modelling, which may strongly affect the selection of coupling strategies, is the large disparity between the fluid convection and solid diffusion time scales [74].

Solid characteristic time

Let us consider a solid with constant properties, for the sake of simplicity, the energy equation of the solid is:

$$\frac{\partial T}{\partial t} = \alpha_s \nabla^2 T \quad (3.5)$$

which has to be solved with proper boundary conditions. Let α_s be the thermal diffusivity of the solid, L_c its characteristic length, and T_0 , the initial temperature of the solid. If the temperature of the solid boundary is fixed to T_{w0} , the characteristics diffusion time that needed by the solid to reach a uniform temperature,

$$t_d \sim \frac{L_c^2}{\alpha_s} \quad (3.6)$$

If we assume that the solid is wetted by a fluid stream with a bulk temperature T_∞ and that the characteristic heat transfer coefficient between the solid and the stream is, h , the characteristic time to heat or cool the solid by the stream is

$$t_c \sim \frac{\rho_s c_s L_c}{h} \quad (3.7)$$

if $t_d \ll t_c$ and it can be assumed that the temperature is spatially uniform, $T = T(t)$. The ratio between both times is called the Biot number,

$$Bi = \frac{t_d}{t_c} = \frac{h L_c}{k_s} \quad (3.8)$$

The evolution time of the solid, t_{solid} , is the maximum of both, $t_{solid} = \max(t_d, t_c)$. If the heat transfer rate at the surface is small ($Bi \ll 1$), $t_{solid} \sim t_c$, whereas if the heat transfer rate is high ($Bi \gg 1$), the bottleneck of the heating process is the solid diffusion and then $t_{solid} \sim t_d$. Biot numbers are usually of order one in realistic applications. The thermal diffusion time assuming that $L_c \sim 10^{-2}m$ and that $\alpha_s \sim 10^{-6} - 10^{-5}m^2/s$ is of the order of $t_d \sim 10 - 100s$.

Fluid characteristic time

The three types of flow in turbomachinery problems are discussed individually.

External flow [74] : Consider an LPT or HPT airfoils operating at high subsonic or transonic conditions. The characteristic velocity $u_c \sim 10^2 m/s$ while the characteristic length $L_c \sim 10^{-2} m$, therefore the through-flow or residence time is,

$$t_r = \frac{L_c}{u_c} \sim 10^{-4} s \quad (3.9)$$

Internal cooling flow : The through-flow of the cooling flow in a cooling passage can be estimated analogously. The Reynolds number, based on the hydraulic diameter of the channel, of cooling flow through an HPT is typically $Re_{D_h} \sim 10^4$, and the passage length, L_c , is, thus the residence time in cooling channels can be estimated as,

$$t_r = \frac{L_c}{u_c} = \frac{L_c D_h}{\nu Re_{D_h}} \sim \frac{10^{-1} \times 10^{-2}}{10^{-5} \times 10^4} \sim 10^{-2} s \quad (3.10)$$

Rotor-Stator Cavity : In the rotor-stator cavity of a gas turbine, the flow is driven by the rotating wall, therefore the flow velocity in the cavity is much lower than the above two cases. It can be demonstrated [75, 76] that the non-dimensionalised characteristic time in rotor-stator cavity, $\tau = \Omega t$, scales as $\tau_c \sim Re_s^{1/2}$, where Re_s is the Reynolds number based on the cavity gap, s , and the rotational velocity, i.e.: $Re_s = \frac{\Omega s^2}{\nu}$, thus

$$t_r \sim \frac{s}{\nu^{1/2}} \sim \frac{10^{-1}}{10^{-5/2}} \sim 10^{3/2} s \quad (3.11)$$

It is apparent that very huge disparity in time scale can exist between the solid and fluid domains, except in the rotor-stator cavity case. This disparity may lead to very stiff problems when performing time-accurate unsteady coupled heat transfer studies. According to the estimation of He and Oldfield [74], in unsteady study, blade passing period time could be a relevant characteristic time in turbomachinery. In turbomachinery typically the Strouhal number or the reduced frequency associated to the blade passing frequency, $St = t_{Blade-passing}/t_r$, is typically of order one, so that no additional characteristic time is added to problem.

The time scale disparity between the solid and the fluid can be as large as 10^6 posing large

difficulties to the direct time accurate simulation of the problem. If for instance we assume that we need 10^2 time steps to compute a through-flow time, the total number of time steps required to solve the problem would be about 10^8 , which is unaffordable for engineering problems.

3.1.2.2 Steady state coupled heat transfer

Some partitioned methods use synchronised time-advancing approaches to form the scheme in fully coupled manner, where both, fluid and solid solvers are marched synchronously in time and exchange information at each prescribed time step to reach steady state. The corresponding numerical stability analysis was earlier studied by Giles using von Neumann stability analysis [77] and further pursued by Heselhause [78]. They stated that the Dirichlet-Neumann interface is a stable choice for coupled heat transfer analysis. A successful implementation can be found in Liu et al. [15]. However, these strongly coupled approaches follow essentially the same strategy as monolithic methods since the time-stepping procedures of the fluid and solid domains are synchronised, therefore are not efficient for practical applications.

In steady state coupled heat transfer problems, as the physical time-accurate intermediate solutions are sought and only the final state is concerned, the time consistency in the coupling scheme is no longer important. As a consequence, any coupling strategy should work as long as the temperature and heat flux continuity condition is satisfied at the final state. Errera and Chemin [73] have proposed an analysis of Dirichlet-Robin interface and more general Robin-Robin interface based on the Godunov-Ryabenkii method [79] and provide a formulated expression of optimal coupling coefficient in terms of convergence speed and stability. In their loosely coupled approach, the solid solver always run till steady state at each fluid time step, and fluid solver is marched in time to reach steady state [80]. However, the drawback is that the fluid solver is still solved in a time accurate manner which is not efficient to solve steady-state problems and could be unaffordable for complex engineering applications. In these cases, the use of fluid solvers featuring mechanisms to accelerate the convergence to the steady state such as preconditioning, local time step, multigrid, implicit methods, or a combination of all of them, is mandatory in practise.

An alternative kind of loosely coupled approaches, where the two solvers are independently converged until reaching the steady state, have been proposed for engineering-oriented works [70, 13, 15, 16]. These methods treat solvers as “blackbox” that deliver a response function independent of the physical time. In this approach attention has only to be paid to the coupling strategy itself, and therefore it has more flexibility to handle any combination of codes, regardless of their resolution methods. It is worth noting that the thermal solvers usually involve much more than simply a solver for the heat diffusion equation, and feature radiation under different forms, reduced order models for convection, HTC correlations, or even coupling with mechanical methods to determine the structure displacements. The re-usability of existing methods is mandatory in practise, and very attractive from an engineering point of view.

3.1.2.3 Unsteady/Transient coupled heat transfer

Furthermore, loosely coupled approaches can also be easily extended to transient analysis based on a “quasi-dynamic” scheme, whereby time derivatives are retained only in the solid domain, whereas the fluid domain is treated as quasi-stationary and temporal derivatives are neglected. The main underlying hypothesis is that the time scale of the fluid domain is much faster than the diffusion time of the solid. This approach allows to perform a complete engine transient analysis with a characteristic time of thousand of seconds. Sun et al. [17, 81] performed coupling analyses with the Dirichlet-Neumann interface for practical engine transient cycle simulations, and Zhang et al. [82] applied this technique for a hypersonic flow case. The description of Robin-Dirichlet interface approach can be found in the work of Errera and Baque [83], Baque et al. [84], Gimenez et al. [85] and Chaquet et al. [86, 87].

Additional to the “quasi-dynamic” approaches, a notable method is proposed by He and Oldfield [74], in which a time-domain fluid solution and frequency-domain solid conduction is adopted in conjunction with a continuously updated Fourier transform at the interface. This method enables the unsteady coupling to be achieved without simultaneously solving the unsteady temperature field in the solid domain, providing to a new prospect of unsteady coupled heat transfer modelling.

3.1.2.4 Motivation

The objective of the present work focuses on improving the computational efficiency of steady-state coupled heat transfer problem. As stated, the main disadvantage of this kind of loosely coupled methods is that for steady state analysis, the CFD solver has to be invoked a few times in an iterative manner leading to relatively large computing times compared to that of an uncoupled stand-alone CFD simulation. This aforementioned iterations is somewhat avoided when fluid solvers are marched in time and thermal solvers converged in each fluid time step (see for instance Errera and Duchaine [80]) but the cost of this type of simulation is much larger than that of a standalone CFD analysis. To the author's knowledge, the quantification of the computational efficiency of loosely coupled methods for steady state problems has been rarely investigated yet.

This work presents a computationally efficient loosely coupled strategy with a basic theoretical support. Realistic complex turbomachinery cases are used as application examples. First, a stability analysis was carried out for a one-dimensional model problem and the influence of the incomplete convergence of the fluid solver is discussed. Then, a more efficient and robust coupling method is proposed according to the results of the analysis. Finally, an impingement cooling passage model and a stator well cavity are used to rank different methods in realistic turbomachinery applications and to verify the validity of the theory using different numerical settings.

3.2 Baseline Loosely Coupled Strategy

Baseline Flow Solver The three-dimensional Navier-Stokes equations in conservative form for an arbitrary control volume may be written in compact form as,

$$\frac{d}{dt} \int_{\Omega} \mathbf{U} d\Omega + \int_{\Sigma} \mathbf{F}(\mathbf{U}) d\mathbf{A} = 0 \quad (3.12)$$

where $\mathbf{U}$ is the vector of conservative variables, $\mathbf{F}$ the sum of the inviscid and viscous fluxes, Ω the flow domain, S its boundary and $d\mathbf{A}$ the differential area pointing outward to the boundary. The flow solver is based on an in-house implementation described by Corral et al. [88], known as Mu^2s^2T . It uses hybrid unstructured grids to discretise the spatial domain and may contain cells with an arbitrary number of faces. The solution vector is stored at the vertexes of the cells. The control volume associated to a node is formed by connecting the median dual of the cells that surround it, using an edge-based data structure. The preconditioned semi-discrete version of non-linear equations for each internal node i of the mesh can be written using a finite volume approach as,

$$P_{Ji}^{-1} \frac{d(\Omega_i U_i)}{dt} + \sum_{j=1}^{n_{edges}} F_{ij} \cdot A_{ij} - D_{ij} = S(U_i) \quad (3.13)$$

where Ω_i is the control volume, A_{ij} is the area associated to the edge ij , n_{edges} is the number of edges that surround the node i , F_{ij} represents the inviscid and viscous fluxes through area A_{ij} , D_{ij} are the artificial dissipation terms, P_J^{-1} is the block-Jacobi preconditioning matrix, S is the source term. The contributions of the domain boundary faces are omitted for simplicity. Turbulence effects are accounted for using the $k - \omega$ turbulence model [89]. The code is parallelized using MPI and executed in GPUs to reduce the turn around time [90]. Further details of the method and current developments may be found in [91, 92].

Baseline Thermal Solver The in-house thermal solver *Mephisto* [93], solves the Heat Diffusion equation in the solids

$$\int_{\Omega} \rho C \frac{dT}{dt} = \int_{\Sigma} k_s \nabla T d\mathbf{A} + \int_{\Omega} Q d\Omega \quad (3.14)$$

where $\mathbf{q}$ is the heat flux at the boundary and Q represents the volumetric heat sources. The spatial discretization is the same as that of the CFD solver. The set of equations are solved using a SOR method while transients are marched in time using a Crank-Nicholson time-stepping scheme.

Fluid-Solid Interface A Dirichlet-Robin interface, in which the wall temperature is imposed in the fluid solver and convective heat transfer boundary conditions are used for the solid thermal solver, has been implemented. The CFD solver first imposes the wall temperature obtained by the heat conduction code in the previous iteration, and solves the fluid problem to obtain the fluid temperature in the whole domain, T_f^n , and the heat flux at the wall, q_f^n . Then, heat conduction is solved for the solid using the following convective boundary condition in the walls in contact with the fluid,

$$q_s^n = h^*(T_{w-solid}^n - T_{ref}^n) \quad (3.15)$$

where h^* is the so-called virtual, artificial heat transfer coefficient, which needs to be specified, and from the point of view of the simulation is considered as a free numerical parameter. The reference temperature, T_{ref}^n , is calculated using the information provided by the fluid solver as,

$$T_{ref}^n = T_{w-fluid}^n - \frac{q_f^n}{h^*}. \quad (3.16)$$

The solution of the aforementioned problem delivers, the whole temperature field in the solid, T_s^n , from which the wall temperatures, $T_{w-solid}^n$, are just a subset. This process is repeated until convergence. Combining (Equation 3.15) and (Equation 3.16), the following expression is obtained,

$$q_s^n - q_f^n = h^*(T_{w-fluid}^{n+1} - T_{w-fluid}^n). \quad (3.17)$$

This equation shows that when the converged state is obtained, and the wall temperature difference between the solid and the fluid is low enough, the flux continuity condition is also satisfied. The final solution is independent of the actual value of h^* , but has to be given as a computational parameter of the numerical scheme. The actual choice of h^* does not modify the final solution. It just alters the convergence history of the scheme and plays the role of a relaxation parameter of the fixed point iteration that drives the process.

Figure 3.3: One-dimensional coupled fluid/solid heat transfer problem

3.3 Analysis of Loosely Coupled Approach with Dirichlet-Robin Interface

There are two strategies to minimise the computational time in this kind of loosely coupled analysis. The first is to reduce the number of sequential coupling iterations between the two solvers. The second is to reduce the evaluation cost of the fluid solver, since in practise the computational cost of the thermal solver is negligible compared to that of the fluid solver. The evaluation cost of the fluid part of the problem depends on the details of the numerics of the fluid solver and will not be discussed here. In this section we focus on the coupling process itself. In the following lines the Dirichlet-Robin approach is discussed.

Consider the one-dimensional steady state heat conduction problem described in Figure 3.3. Assume that the characteristic length in the longitudinal direction, is much larger than the plate thickness, L , and then heat is mainly transferred by conduction only along the x direction. The temperature at the bottom of the solid, T_0 , is specified and k_s is the thermal conductivity of the solid. In the fluid side, T_{ref} represents the local reference temperature against which heat is exchanged, T_w is the wall temperature and q is the heat flux at the wall.

In our model problem, the thermal response of Navier-Stokes equations is expressed using the Newton's law as

$$q_f = h(T_w - T_{ref}), \quad (3.18)$$

where h , is the physical heat transfer coefficient, whereas the exact solution of the heat diffusion

equation in the solid domain is given by

$$q_s = \frac{k}{L}(T_0 - T_w). \quad (3.19)$$

(Equation 3.18) and (Equation 3.19) can be considered as the simplified response of the fluid and solid “black-boxes” to the inputs. The exact solution, T_w , to this problem is obtained imposing that $q_f = q_s$,

$$\frac{T_w - T_{ref}}{T_0 - T_{ref}} = \frac{1}{1 + Bi} \quad (3.20)$$

where $Bi = hL/k$ is the Biot number.

3.3.1 Fully Converged CFD

Let us suppose that the wall temperature at the n^{th} iteration in the fluid, $T_{w,f}^n$, differs e_T^n , from the exact solution of the problem, T_w , or in other words, e_T^n , is the error of the solution at the n^{th} iteration,

$$T_{w,f}^n = T_w + e_T^n \quad (3.21)$$

According to the iteration scheme outlined in the previous section, the thermal response of Navier-Stokes equations at the the n^{th} iteration can be expressed as $q_f^n = h(T_w^n - T_{ref})$, where both, h and T_{ref} , are unknown in practise, since only the response of the Navier-Stokes solver, q_f^n , is known. The reference temperature is unknown as well in complex problems, where the fluid temperature outside the boundary layer changes along the boundary, and cannot be determined. Then we derive T_{ref}^n as,

$$T_{ref}^n = T_w + e_T^n - \frac{q_f^n}{h^*}, \quad (3.22)$$

which delivers the same heat flux but for h^* , instead of h . The convective (Robin) boundary condition for the heat diffusion in the solid model is introduced using the virtual HTC h^* ,

$$q_f^n = h^*(T_{w,s}^n - T_{ref}^n) \quad (3.23)$$

since the physical HTC, h , is unknown also. The solution of the solid part yields,

$$q_s^n = \frac{k}{L}(T_0 - T_{w,s}^n), \quad (3.24)$$

and imposing that

$$q_f^n = q_s^n, \quad (3.25)$$

we obtain the new temperature in the solid, $T_{w,s}^n$, that becomes the new temperature for the fluid, $T_{w,f}^{n+1} = T_{w,s}^n$. After some algebraic manipulations we can derive an expression that relates the errors between two consecutive iterations,

$$e_T^{n+1} = \frac{Bi^* - Bi}{1 + Bi^*} e_T^n \quad (3.26)$$

where $Bi = hL/k_s$ and $Bi^* = h^*L/k_s$ are, respectively the physical and virtual Biot numbers. We can define the iteration amplification factor, G^* , as $G^* = e_T^{n+1}/e_T^n$, to obtain:

$$G^* = \frac{Bi^* - Bi}{1 + Bi^*} \quad (3.27)$$

This expression indicates that the Dirichlet-Robin approach is unconditionally stable when physical Biot number, $Bi \leq 1$, and stable if $Bi^* > \frac{Bi-1}{2}$ when $Bi > 1$. The convergence and the stability region of the method are sketched in Figure 3.4 left and right, respectively. Expression (Equation 3.27) shows that when $Bi^* \Rightarrow Bi$, $G^* \Rightarrow 0$ and the convergence of the method is very fast.

Since the stability condition is equivalent to $h^* > \frac{h}{2}$, the numerical scheme is controlled in practise specifying h^* . The closer h^* is to h , the faster the convergence rate. In addition, if the estimate of h^* is close to the physical HTC, no stability problems will be encountered. This explains why Dirichlet-Robin approach is more stable than others and why large values of h^* are used in practise to ensure stability. This kind of analysis and a summary of the results for other approaches can firstly be found in [72].

Due to the spatial disparity of the physical HTC in a real application local virtual HTCs are very

Figure 3.4: Dirichlet-Robin approach convergence sketch (left) and stability region (right)

convenient to speed-up the convergence of the method. If a single value of Bi^* , or h^* , is used, we need large values of Bi^* in regions of high physical HTC. This is why existing experiences with the Dirichlet-Robin approach recommend the use of large constant positive values of h^* for all the nodes to avoid stability problems [16, 94]. This choice ensures stability, but it is detrimental for the convergence in regions where Bi is small, or of order unity, since in these zones $|G^*| \Rightarrow 1$, and the convergence rate is very poor. Moreover, the main issues usually take place during the initial stages of the simulations, when the variations from iteration to iteration are large, hence the need to dynamically update the h^* value.

3.3.2 Incompletely Converged CFD

Computational cost can be further reduced in loosely coupled analysis by minimising the evaluation cost of the fluid solver, which is in practice much greater than that of the thermal solver, unless complex and expensive radiation models are included. This objective can be achieved, for a given solver, by reducing the convergence tolerance during the intermediate coupling iterations, where the exact solution of the flow-field is not actually required.

The Navier-Stokes equations using a local time t stepping for local iterations can be written in compact form as,

$$\frac{\partial \mathbf{U}}{\partial t} = \mathbf{R}(\mathbf{U}) \quad (3.28)$$

where the steady state solution is given by $\mathbf{R}(\mathbf{U}^E) = 0$, and $\mathbf{U}^E$ is the the exact steady solution.

If the solution is not fully converged then,

$$\mathbf{R}(\mathbf{U}) = \mathbf{R}(\mathbf{U}^E) + \left(\frac{\partial \mathbf{R}}{\partial \mathbf{U}} \right)_E (\mathbf{U} - \mathbf{U}^E) = \mathbf{R}(\mathbf{U}^E) + \boldsymbol{\varepsilon}^R \quad (3.29)$$

where $\boldsymbol{\varepsilon}^R$ is the error in the residuals, $\mathbf{R}(\mathbf{U}^E) = 0$ and $\boldsymbol{\varepsilon}^U = \left(\frac{\partial \mathbf{R}}{\partial \mathbf{U}} \right)_E^{-1} \boldsymbol{\varepsilon}^R$ is the solution error.

We have mentioned that in our simplified model problem, the exact solution of the fluid solver for a given wall temperature, $T_{w,f}^n$, is represented by $q_f^n = h(T_{w,f}^n - T_{ref})$. If the solution is not fully converged an error is induced in the heat flux, and can be written as,

$$q_f^{n*} = (h + \Delta h^n)(T_{w,f}^n - T_{ref}) \quad (3.30)$$

where Δh^n represents the error in the physical HTC, and hence in the heat flux, in the n^{th} iteration, and the asterisk is used to denote that the variable has an error induced by the incomplete solution of the fluid solver. It is necessary to recall here that neither h , nor Δh^n , are actually known in practice. Although the existence of HTC in a transient or pseudo-transient analysis is arguable, we have adopted here this nomenclature to make it consistent with that of the previous section.

Following the same scheme analysis as in the previous section and using (Equation 3.22) and (Equation 3.23) we obtain,

$$T_{ref}^{n*} = T_{ref}^n - \frac{\Delta h^n}{h^*}(T_{w,f}^n - T_{ref})$$

This expression is then injected in (Equation 3.25) to solve the solid problem. Neglecting second order terms, we obtain the equivalent expression to (Equation 3.26), namely

$$e_T^{n+1} = G e_T^n + \frac{\Delta Bi^n}{1 + Bi^*}(T_w - T_{ref}) \quad (3.31)$$

where $\Delta Bi^n = \Delta h^n L/k$, is the non-dimensional error induced by the incomplete solution of the fluid solver. It is readily seen from (Equation 3.31) that, even if $\|G\| < 1$, the exact solution can

not be obtained below a certain threshold, e_T^∞ , if the fluid solver is not fully converged,

$$\frac{e_T^\infty}{(T_w - T_{ref})} = \frac{\Delta Bi^\infty}{(1 + Bi)} \quad (3.32)$$

where ΔBi^∞ is the asymptotic convergence of the CFD solver error. The convergence of the fixed point iteration method is outlined in Figure 3.5. It can be seen that during an initial phase of the convergence process, where it is assumed that the incomplete solution of the fluid solver yields a constant error, ΔBi^n , the solution tends to converge to a different solution and therefore and asymptotic error, e_T^∞ , is obtained. This means that the final solution is as accurate as the solution of the CFD solver is. In practise fluid solvers are usually initialised from the previous iteration, and usually

$$\lim_{n \rightarrow \infty} \Delta Bi^n \rightarrow 0. \quad (3.33)$$

If the previous condition is not fulfilled, the final solution contains a systematic error, as it has been previously mentioned. If the systematic error due to the incomplete convergence of the fluid solver, ΔBi^n , is constant, a simple analysis of (Equation 3.31) leads to the conclusion that the convergence rates of the complete and incomplete CFD solution approaches are the same. For this reason, the curve corresponding to the incomplete solution of the CFD (dot-dashed lined in Figure 3.5) has been drawn parallel to that of the fully converged scheme (dashed line in the same figure). Usually the systematic error associated with the incomplete resolution of the Navier-Stokes equations decreases with the number of fixed point iterations, since the initial solution of the fluid solver is much closer to the final solution during the last iterations of the convergence process than during the first ones. This dependence of the error, ΔBi^n , on that of the previous iteration, $n - 1$, can induce instabilities in the convergence process. It can be shown that, during the initial stages of the process, when the errors are large, a minimum convergence level of the fluid solver has to be imposed, to ensure the overall convergence of the method.

Figure 3.5: Error sequence of the incompletely converged CFD scheme. Inset: Converge history for different CFD strategies.

3.3.3 Discussion on the Convergence Level

The whole process fails to converge if the convergence level of the CFD is too low. The underlying reason is that if the thermal boundary layer is not well resolved the computed heat flux during the pseudo-transient can be even of opposite sign of the actual one, potentially giving rise to an instability. Let us consider for example a flat plate immersed in free stream, let us assume that the plate is hotter than the free steam, and that, suddenly, the temperature difference between both is halved. In the final state the heat flux between the plate and the fluid will be approximately the half, but there is a transient, of the order of the diffusion time of the thermal boundary layer, where the heat flux has a different order of magnitude, and even the opposite sign, from the final one. If the fluid solver is stopped before the thermal boundary layers is approximately converged, the intermediate results quite likely can lead to a numerical or pseudo-physical instability. This problem was addressed in using some simplifying hypotheses in [73].

Complex engineering problems usually have more than one characteristic time. If the second characteristic time is much longer than the diffusion time of the thermal boundary layer, a smart strategy is to decouple both scales using incompletely converged CFD simulations, where the global problem is not converged, but the boundary layers are converged in a quasi-stationary sense. When dealing with this type of problems, the use of time-accurate approaches is unaffordable, and accelerated steady solvers that make use of preconditioning, multigrid, implicit methods, or a combination of all of them, is compulsory. In this context the analysis of the convergence of the whole process is further complicated.

The instability mechanism associated to the incomplete convergence of the CFD solver is implicit in (Equation 3.31), but it has not been properly exploited. If ΔBi^n is constant, i.e.: it does not depend on the iteration number, the convergence criterion reduces to that obtained for fully converged fluid solutions, namely, $\|G\| < 1$. The stability analysis previously outlined is based on the hypothesis that the heat flux can be expressed as $q = h(T_w - T_\infty)$, which is essentially a steady approach. If we approximately assimilate the incomplete resolution of the fluid solver, to an unsteady problem, then the actual heat flux is composed of a steady component, essentially

the Newton's law, and an unsteady part, q_u , which has the form;

$$q_u = \frac{k_f}{\delta_T} (T_w - T_w^i) c'(\tau) \quad (3.34)$$

where, T_w^i is the initial temperature at the wall, and $c'(\tau)$ is a monotonically decreasing response function of the non-dimensional time, $\tau = t\alpha_f/\delta_T^2$, being α_f , the thermal diffusivity of the fluid, and δ_T the thermal boundary layer thickness. In general $c'(\tau) \ll 1$ for $\tau \gg 1$. If we associate T_w^i to the wall temperature in the previous iteration, $T_w^i = T_w^{n-1}$, we can use this expression to estimate the non-dimensional heat flux error due to the incomplete convergence of the fluid solver, ΔBi^n , as:

$$\Delta Bi^n = c(\tau) \frac{\Delta T^n - \Delta T^{n-1}}{T_w - T_\infty} \quad (3.35)$$

where $c(\tau)$ is,

$$c(\tau) = -\frac{k_f}{k_s} \frac{L}{\delta_T} c'(\tau). \quad (3.36)$$

Inserting expression (Equation 3.35) in (Equation 3.31), it is possible to obtain the following expression:

$$G^2 - G(G^* + c) + c = 0 \quad (3.37)$$

which is a second order equation for G , since now, on the contrary to fully converged case, the temperatures at three different iteration levels are linked, because ΔBi^n retains some history effects. Obviously if the fluid solver is fully converged, $c = 0$, and $G = G^*$. It can be shown that the condition $|G| = 1$ is obtained when $c = -(1 + G^*)/2$, and therefore, since $|G^*| < 1$, it is enough that $|c| < 1$ to ensure the convergence of the scheme. This in practice forces the fluid simulation to run for a minimum pseudo time or a minimum number of iterations. This simplified qualitative analysis is interesting because it reveals the nature of the problem.

In this work we have tackled this problem partially converging the fluid solver during the first $n_{switch} - 1$ iterations, (typically two orders of magnitude less than in the fully converged case), whereas in a second phase, the flow is fully converged to remove systematic errors from the solution (see Figure 3.5). This approach has shown to be robust and efficient.

Figure 3.6: Scheme of dynamic virtual HTC approach

3.3.4 Implementation Details

Standard Dirichlet-Robin coupling approaches use large and constant values of the virtual HTC in the whole model to ensure stability. However, we have shown that h^* should be chosen as close as possible to the physical HTC, h , in order to obtain fast convergence rates while ensuring the stability of the method. The difficulty is that in realistic applications, physical HTCs are far from uniform and exhibit large spatial variations. Finding a proper h^* distribution, before the simulation is even performed, is a critical problem in itself. Hereby we propose a dynamic updating of the virtual HTC which is automatically computed as a part of the coupling process. In addition, the use of incompletely converged CFD is also considered to further speed up the coupling process. The implementation of the new scheme is described in the following lines.

3.3.4.1 Dynamic virtual HTC

A large constant value of h^* is arbitrarily selected for the first two iterations. Then the virtual HTC at iteration $n+1$ is computed resorting to the Newton's law ($q = h(T_w - T_{ref})$) and the use of two consecutive iterations,

$$h_i^{*,n+1} \simeq h_i = \left(\frac{dq}{dT_w} \right)_i \simeq \left| \frac{q_i^n - q_i^{n-1}}{T_{w,i}^n - T_{w,i}^{n-1}} \right| \quad (3.38)$$

where it is obvious that as the solution converges, $h^* \rightarrow h$. The updating stops when the local wall temperature variation at two consecutive iterations is smaller than a specified threshold $|T_{w,i}^n - T_{w,i}^{n-1}| < \beta$, typically 3K, to minimise the noise associated to the numerical evaluation of dq/dT_w . At this point, we consider that h_i^* is close enough to h to be frozen for the calculation.

Limitations to the local maximum and minimum h^* are set to avoid meaningless values for h^* during the iteration process. The objective here is not to obtain an accurate value of the HTC, but (i) to find a reasonable distribution to speed up the coupling scheme and (ii) free the user from the burden of choosing h^* by trial and error.

3.3.4.2 Use of incompletely converged CFD

When incompletely converged CFD solutions are used, the convergence level during the first few iterations, where the error is relatively large, is low to reduce the computational cost. After a few iterations, the scheme switches to the use of fully converged solutions. Adopting this idea, the following scheme is proposed:

1. When $n < n_{switch}$, CFD stops when the the residual of the incompletely converged solution: $R = R_{int} = 10^2 R_c$, where R_c is the residual of the fully converged solution (see inset of Figure 3.5)
2. When $n \geq n_{switch}$, in every CFD iteration the fluid solver is run until fully convergence and $R = R_c$, with this scheme, the systematic error vanishes and $e_T^n \rightarrow 0$.

As shown in (Equation 3.31), the residual term R is determined by the CFD residual, coupling error, and system itself. In a particular problem there is an optimal value that delivers the smallest computational time. However, this value may not be suitable for general applications. A more conservative value must be used because of robustness. Our current experience is to set the CFD stopping criterion as $R_{int} = 10^2 R_c$ and the iteration for switching to $n_{switch} = 5$. This approach is robust enough for engineering applications. Finally, the strategy can be summarised in Figure 3.6. The main comment is that the virtual HTC of the previous iteration, h_n^* , is used to estimate the virtual HTC of the next iteration.

Figure 3.7: Computational domain and mesh for the coupled heat transfer problem of laminar air flow over a finite thickness flat plate

3.3.5 Validation

The objective of this subsection is to validate the coupling method with a simple and well studied conjugate heat transfer problem. With this objective in mind we have chosen a test case, the heat transfer problem between a flat plate and a laminar boundary layer of air ($Pr=0.7$) flowing over it (see Figure 3.7). The free stream inlet Mach number is $Ma = 0.2$, so that the flow can be considered incompressible, and the Reynolds number based on the plate length, L , is $Re_L = 4.55 \times 10^5$. The actual values of the free stream static pressure and temperature correspond to standard ambient conditions, $p_\infty = 10^5$ Pa and $T_\infty = 20^\circ$ C, respectively. A constant temperature, $T_b - T_\infty = 100K$, is imposed at the bottom surface of the plate, while adiabatic conditions are used at the two side walls. The upper wall of the plate is coupled with fluid domain. The plate thickness is $l/L=0.05$, and different solid to fluid thermal conductivity ratios, k_s/k_f , have been simulated to cover a wide range of materials. Namely the cases $k_s/k_f=40, 400$ and 4000 have been checked, which correspond to mean Biot numbers based on the plate thickness of $Bi = 0.5, 0.05$ and 0.005 , respectively.

A structured mesh of 101×60 , points has been generated for the fluid domain. The point distribution is highly clustered near the wall region to ensure sufficient spatial resolution ($y^+ < 1$). The solid domain uses an unstructured grid which is conformal with the fluid grid at the solid-fluid interface. Numerical results have been compared against the analytical results of the same conjugate convective heat transfer problem derived by Luikov [95] and a more recent work of Lindstedt and Karvinen [96]. Figure 3.8 compares the wall temperature distribution along the plate between the numerical and the quasi-analytical solution. It can be seen that the

Figure 3.8: Coupled wall temperature distribution along a flat plate for different k_s/k_f ratios. Inset: Span-wise temperature distribution at the flat plate mid-point

numerical results predict quite well the analytical solution for the whole range of k_s/k_f .

It can be seen that the span-wise temperature distribution in the mid-point of the plate (insets of Figure 3.8) exhibits a linear region in the solid part and high thermal gradient in the thermal boundary layer of the fluid. The temperature gradient in the solid is higher for the case with the lowest conductivity ($k_s/k_f = 40$) than for the case with highest ($k_s/k_f = 4000$), where the temperature profile in flat plate span-wise direction is nearly constant, as expected.

3.4 Practical Case Study

Two representative configurations of turbomachinery applications have been used as a vehicle to test the computational efficiency and robustness of the proposed method in realistic 3D cases. The models are described in the following sub-sections, however due to their complexity it is not possible to define a unique characteristic length and a single characteristic heat transfer coefficient. It can be said that the local Biot number in both configurations oscillates between nearly 0 to 1.5.

3.4.1 Leading Edge Impingement Cooling Channel

A simple leading edge impingement cooling model is shown in Figure 3.9. The model consists of two flow channels, connected through ten staggered impingement cooling holes. The coolant flow enters from the root of the mid-chord passage and then pass through the impingement hole to cool the leading wall of the front passage. The flow leaves from the tip of the front passage. The solid model is located at the leading edge and coupled with the fluid domain through the three surfaces of the front channel to investigate the coupled heat transfer effects. The rest of the surfaces are assumed to be adiabatic.

Both the fluid and solid meshes have been created using the same in-house grid generator using tetrahedral elements. Prismatic layers are generated in the fluid domain wall before the creation of the volumetric grid. The computational model has around 0.6 million and 46 thousand cells

Figure 3.9: Leading edge impingement cooling channel configuration for coupled heat transfer analysis

in the fluid and solid domains respectively. The inlet total pressure is 130k Pa, total temperature is 600K and the inlet turbulence level is 3%. The leading surface of the solid model is set to a constant temperature of 1150K. Wilcox's $k - \omega$ turbulence model [89] is used to account for turbulent effects. The estimated characteristic Biot number for this case is about $0 \sim 1.5$.

3.4.2 Turbine Stator Well Cavity

Figure 3.10 displays the geometry of a turbine stator and the associated front and rear stator well cavities. The 3D coupled heat transfer analysis uses a single pitch of the stator wheel, and the disk holes have been replaced by a slot of equivalent area. The stator blade is coupled with the fluid domain, while the temperature distribution along the rotor parts is taken from experimental data [97]. A comprehensive set of simulations of this configuration can be found in [98, 99].

The main flow path has inlet total temperature of about 385K, the inlet mass-flow is 4.8 kg/s and the total pressure is 3 bars. The temperature of the cooling flow is around 350K. The static

Figure 3.10: Left: Stator well cavity configuration for coupled heat transfer analysis. Right: Computational domain and measurement points

Baseline approach	Case 1: constant h^*	$h^* = 0.1 \text{ mW/mm}^2 \text{K}$
	Case 2: constant h^*	$h^* = 1 \text{ mW/mm}^2 \text{K}$
	Case 3: constant h^*	$h^* = 3 \text{ mW/mm}^2 \text{K}$
Dynamic approach	Case 4: dynamic h^*	CFD fully converged
	Case 5: dynamic h^*	CFD incompletely converged ($n_{\text{switch}} = 5$)

Table 3.1: Test matrix for coupled heat transfer analysis

pressure at the main flow exit and the inlet total pressure of the cooling flow are adjusted to match the measured mass flows. The Wilcox's $k - \omega$ turbulence model [89] is used as well in this simulation. The grid size of the CFD model is about 4 million cells, while the solid model of the stator contains 30 thousand cells. The estimated Biot numbers for different regions of the model are the following: stator hub, $Bi \simeq 0 \sim 1.3$, stator tip, $Bi \simeq 0 \sim 1.5$, and stator vane $Bi \simeq 0 \sim 0.5$.

3.4.3 Computational Test Matrix

The two configurations have been simulated using five different numerical setups to compare their convergence behaviour and computational time. The simulation plan is shown in Table 3.1. Cases 1 to 3 use the baseline approach with three fixed values of the virtual HTC, $h^* = 0.1$, 1 and $3 \text{ mW/mm}^2 \text{K}$. Cases 4 and 5 use the proposed dynamic virtual HTC approach where

Figure 3.11: Impingement cooling channel coupling configuration wall temperature, in K , (left) and HTC, in mW/mm^2K , (right) distributions

h^* is locally computed and dynamically changed from iteration to iteration. The difference between the latter two cases is that while Case 4 uses fully converged CFD solutions, Case 5 employs incompletely converged solutions. As it has been previously explained, incompletely converged CFD solutions are employed in the first four coupling iterations. Both Cases 4 and 5 use $h^* = 1mW/mm^2K$ for the first two iterations since the h^* distribution is unknown at the beginning.

3.5 Results and Discussion

Figure 3.11 and Figure 3.12 show the coupling wall temperature and heat transfer coefficient contour plots of the two configurations. All the five numerical setups provide exactly the same predictions for the temperature field, because in the Dirichlet-Robin approach, the actual value of the virtual HTC does not influence the final result (Equation 3.17).

In the stator well cavity model, numerical and experimental temperatures results are compared at the measurement points (see Figure 3.10 right) for both, the coupled and the uncoupled meth-

Figure 3.12: Stator well cavity configuration coupling surface temperature in K (left) and HTC in mW/mm^2K (right) distributions

Figure 3.13: Relative error of coupled heat transfer method and uncoupled method at measurement points

ods. Figure 3.13 shows that the coupled heat transfer method provides more accurate results than the uncoupled approach. The main purpose of Figure 3.13 is to gain confidence in the overall framework on which the coupling method has been implemented, since the discussion of the coupling results is not the objective of this work.

Figure 3.14: Convergence history for the impingement cooling channel configuration

3.5.1 Convergence Behaviour

The convergence history plots of the two configurations as a function of the number of coupling iterations are presented in Figure 3.14 and Figure 3.15. The relative error to monitor the convergence, ε , is defined as,

$$\varepsilon = \frac{1}{(T_{hot} - T_{cold})} \sqrt{\frac{1}{N} \sum_{i=1}^N (T_{w,i}^n - T_{w,i}^{n-1})^2} \quad (3.39)$$

where N is the number of boundary nodes at the coupling interface, and T_{hot} and T_{cold} are, respectively, the maximum and minimum wall temperatures of the fluid and solid domains.

According to (Equation 3.27), the slope of the convergence curve for the baseline Dirichlet-Robin approach (Cases 1 to 3) should be nearly constant in logarithmic scale. Due to the fact that, the iteration amplification factor G is fairly constant when h^* is fixed and the real HTC distribution does not change significantly with the variation of the wall temperature during the iteration process. In addition, the convergence rate is determined by the differences between h^* and the real HTC. Therefore, Case 2 has faster convergence rate than Case 3, as its h^* is closer to the real HTC mean value (Figure 3.11 and Figure 3.15). However, it is important to notice that, Case 1 failed to converge in both configurations. This is because, at some locations of the

Figure 3.15: Convergence history for the stator well cavity configuration

coupling interface, the local physical Biot number may be larger than 1. As a consequence, if h^* is smaller than half of the actual HTC, the convergence criterion is no longer fulfilled. This lack of local convergence in complex 3D cases can only be seen in the convergence history when the relative error is smaller than a certain threshold. This is the reason why, in practical engineering applications, the conventional Dirichlet-Robin approach is always recommended to be used in conjunction with large h^* (e.g. $3\text{ mW/mm}^2\text{K}$).

In the dynamic h^* approach, the convergence behaviour at the first two iterations are identical in Cases 2 and 4, since the same h^* is adopted during the initial steps. Case 5 presents some difference due to the use of incomplete convergence CFD solutions.

From the third iteration, when the dynamic evaluation of virtual HTC is activated, the convergence rate increases rapidly as the coupling process advances. The coupling scheme using fully converged CFD solutions converges in fewer iterations than the incompletely converge approach (see Figure 3.14) using both much less fixed point iterations than any of the cases with constant h^* . This means that both dynamic cases, based on a local adaptive evaluation of h^* , can provide an approximate h^* distribution which is closer to the real HTC than the baseline approach and converges faster. Case 4 is the one that requires the fewest number of iterations, because the use of incomplete CFD solutions in Case 5 can not deliver the same level of ac-

Figure 3.16: Convergence history as a function of the computational time for the impingement cooling channel configuration. Every individual symbol represents an iteration

curacy to the HTC estimate as the approach using fully converged CFD solutions. In general, the convergence behaviours of all the numerical results follow the trends predicted by the 1D analysis, which is considered a valuable tool.

3.5.2 Computational Time

The computational cost for the impingement and stator well cavity models is shown in Figure 3.16 and Figure 3.17. For each configuration, the same computer resources and settings are used for all the five cases in order to make the computational time comparable. The computational cost is normalized with the cost of a single standard CFD solution of the same problem in the following way,

$$\tau = \frac{t_{coupling}}{t_{cfd-alone}} \quad (3.40)$$

which is a measure of the computational efficiency of the coupling scheme. The first case (the leading-edge impingement model) has about 0.6M cells and a fully converged single steady CFD solution can be obtained in about 1.5 hours in a single Nvidia GeForce GTX780 Ti GPU. The second case (the stator well cavity model) has about 4.05M cells and a steady CFD

Figure 3.17: Convergence history as a function of the computational time for the stator well cavity configuration. Every individual symbol represents an iteration

solution can be obtained in about 1.8 hours in 8 GPUs running in parallel. These data correspond to $t_{cfd-alone}$ in (Equation 3.40).

The two successfully converged conventional cases, Case 2 and 3 converge in significantly different computational times in both configurations, being those cases whose h^* is closer to the real HTC that converge the fastest. The two dynamic cases spend less than 2.5 times computational time needed to converge than a single stand-alone CFD simulation with the same accuracy as the conventional approach. Moreover, the incomplete CFD approach converges in only 35% more time than a single CFD solution although more coupling iterations are needed.

In all the four cases who use fully converged CFD solutions (Case 1 to 4), most of the computational time was spent during the first few iterations, where the variation of the wall temperature between two consecutive coupling iterations is relatively large, and therefore the CFD scheme needs more time to converge. The main idea of using incompletely converged CFD solutions is that, since intermediate solutions do not have any physical meaning, and the computational cost of the initial iterations is large, but they are still far away from the final solution, the CFD convergence criterion can be relaxed for these iterations to reduce the overall computational

time. However, a proper selection of the convergence level is needed to avoid corrupting the global coupling scheme and the final solution.

For all the cases, the time spent to converge the CFD solutions at the latest stages of the coupling process is very small. Figure 3.16 represents with a symbol every single iteration of the convergence history as a function of the non-dimensional computational time. It can be observed that the latest iterations hardly contribute to the total computational cost because the initial CFD solution for these iterations, is very close to the final converged solution, and the problem is solved in a few number of internal CFD iterations.

In the Dirichlet-Robin approach, the determination of the virtual HTC distribution can be difficult to predict without a very good knowledge of the problem. This is especially true in very complex configurations, such as turbine blade internal cooling channel or turbine disc-cavities, where even the bulk temperatures are difficult to define and determine. User defined constant h^* can give rise to either slow convergence rates (Case 3) or unstable solutions (Case 1). The dynamic virtual HTC approach is not only much faster than the baseline method, but more robust, since the convergence criterion is automatically fulfilled. This robustness is a key element to implement automatic methods without any user inputs.

The dynamic approach is faster than the baseline one for two reasons: First, the disparity of physical HTC values along the fluid-solid interface can be automatically taken into account. This frees the method, or the user, from the selection of a constant virtual HTC that take into account the local characteristics of the different flow regions. In other words, when there are large differences in the HTC distribution, the conventional approach based on the existence of a single virtual HTC induces necessarily a computing penalty or a local instability in the method since it is impossible to find out the optimum solution which is based on a local selection. Second, even in the case that the HTC were constant, the dynamic approach provides a mechanism to find out the optimum virtual HTC during the iteration process. As a result, the dynamic method is about two to three times faster, if fully converged CFD solutions are used, than the baseline method, depending on the cases. If incompletely converged fluid solutions are used this factor can be up to five in accordance with the author's experience.

3.6 Summary

This chapter describes a new approach to reduce the computational time of loosely coupled fluid/solid heat transfer methods for steady state heat transfer analysis. Convergence analyses based on a simple one-dimensional model show that, in the Dirichlet-Robin approach, the convergence behaviour is influenced by the physical Biot number and can be controlled by a user specified virtual HTC. The closer the virtual HTC distribution is to the physical one, the faster the convergence rate. A bias error appears when CFD solutions are not fully converged. Thus, the selection of CFD convergence level has to be considered carefully. In this case, fully converged CFD solutions are required during the last few iterations.

Using this 1D analysis, a new method based on a dynamic updating process for seeking the optimal virtual HTC is proposed. The new method frees the user from the selection of a constant virtual heat transfer coefficient that ideally should take into account the local characteristics of the different flow regions.

Two realistic 3D configurations have been studied to test the performance of the proposed method. It is shown that the 1D stability criterion can be used to predict the convergence behaviour of 3D complex configurations as well. The dynamic virtual HTC approach provides faster convergence than the baseline one, and, what is more important, it is robust. Further reduction in computational time is obtained using incompletely converged CFD solutions during the first few iterations with a proper selection of the CFD convergence level without corrupting the global coupling scheme and final solution. The use of fully converged CFD solutions requires fewer number of iterations than that of incompletely converged solutions but is computationally more intensive. By using the incompletely converged CFD scheme, the extra cost of running a fully coupled solid-fluid heat transfer problem is estimated to be about 35% of the computational cost of the equivalent uncoupled CFD solution.

4 Rotational Effect of Internal Channel Flows

Contents

4.1	Navier-Stokes Equations in Relative Frame of Reference	96
4.2	Relevant Non-Dimensional Parameters in Rotating Channels	97
4.3	Rotational Effects in Internal Cooling Channel	102
4.4	Flow and Heat Transfer in 3D Ribbed Channels	117
4.5	Summary	122

Rotating flows can be found in aero-engines due to the presence of rotating parts for work extraction. These include several types of rotating disk systems, such as shrouded rotor-rotor cavities in the compressor and turbine drums, rotor-stator cavities in turbines, as well as straight channels rotating about axes parallel or an orthogonal the engine shaft. The presence of Coriolis and centrifugal forces in the relative frame of reference has a drastic influence on the flow patterns, which are radically different to that observed in stationary cases. In order to study the heat transfer performance in rotating channels, it is necessary to understand the effect of rotation on the flow. In this chapter we discuss the the flow and the heat transfer of rotating channels whose stream-wise direction is orthogonal to the engine axis, with a radially outward flow, since they represent the simplest model of the internal cooling channel of a gas turbine rotor blade.

4.1 Navier-Stokes Equations in Relative Frame of Reference

In the analysis of flow in a steady rotating system, it is convenient to consider the flow from a relative coordinate system that fixed to the rotating parts. Such relative frame of reference allows us to work with fluid motions which are steady. However, additional attention has to be paid since this rotating frame of reference is not inertial. The Navier-Stokes equations in a rotating frame of reference become following forms.

The continuity equation: The continuity equation in the relative frame of reference is unchanged:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{w}) = 0 \quad (4.1)$$

The momentum equation: The momentum equation in the relative frame of reference reads:

$$\frac{D\mathbf{w}}{Dt} = -\frac{1}{\rho} \nabla p - \underbrace{\frac{2\boldsymbol{\Omega} \times \mathbf{w}}{\rho}}_{\text{Coriolis force}} - \underbrace{\frac{\boldsymbol{\Omega} \times (\boldsymbol{\Omega} \times \mathbf{r})}{\rho}}_{\text{centrifugal force}} + \underbrace{\nabla \cdot \bar{\bar{\boldsymbol{\tau}}}}_{\text{viscous stress}} \quad (4.2)$$

where the forces acting in the inertial frame has to be complemented by adding two additional force terms known as the Coriolis and centrifugal forces.

The energy equation: The total energy equation in relative frame of reference can be written as:

$$\rho \frac{D}{Dt} \left(e + \frac{w^2}{2} - \frac{\Omega^2 r^2}{2} \right) = -\nabla \cdot (p \mathbf{w}) + \nabla \cdot (k \nabla T) + \nabla \cdot (\bar{\bar{\boldsymbol{\tau}}} \cdot \mathbf{w}) + Q \quad (4.3)$$

where e is the internal energy, Q represents volumetric heat source terms.

Meanwhile, the kinetic energy equation in the relative frame of reference is obtained by taking the scalar product of the momentum equation with the relative velocity $\mathbf{w}$, we have

$$\rho \frac{D}{Dt} \left(\frac{w^2}{2} \right) = -\mathbf{w} \cdot \nabla p + \underbrace{\rho \mathbf{w} \cdot \nabla \left(\frac{\Omega^2 r^2}{2} \right)}_{\text{contribution of centrifugal force}} + \mathbf{w} \cdot \nabla \cdot \bar{\bar{\boldsymbol{\tau}}} \quad (4.4)$$

Note that the work of the Coriolis force is null since it is perpendicular to the relative velocity.

The equation for the internal energy is finally obtained subtracting the total and the kinetic energy equation reference reads

$$\rho \frac{De}{Dt} = -p \nabla \cdot \mathbf{w} + \nabla \cdot (k \nabla T) + \mu \Phi + Q \quad (4.5)$$

where $\mu \Phi$ is the viscous dissipation.

In addition, using the continuity equation, the total energy equation Equation 4.3 can be further written as

$$\rho \frac{DI}{Dt} = \frac{\partial p}{\partial t} + \nabla \cdot (k \nabla T) + \nabla \cdot (\bar{\bar{\tau}} \cdot \mathbf{w}) + Q \quad (4.6)$$

The quantity I is termed as rothalpy and it is defined as,

$$I = h + \frac{w^2}{2} - \frac{\Omega^2 r^2}{2} = H - \frac{\Omega^2 r^2}{2} \quad (4.7)$$

where $h = C_p T$ is the enthalpy and H , is the stagnation enthalpy in rotating frame of reference.

4.2 Relevant Non-Dimensional Parameters in Rotating Channels

Figure 4.1 depicts the rotating components in a modern aero-engine. In internal cooling systems of rotor blades, the flow and heat transfer processes have certain characters that may allow us to simplify the problem from general expressions. For instance,

1. The internal cooling flow is single phase flow, with no chemical reactions and can be mostly treated as ideal gas.
2. The geometrical dimensions are much larger than the fluid molecules, so that no-slip conditions at walls can be considered.
3. The Mach number in the channel is very low to prevent large pressure loss.
4. In design process, the cooling channel rotates at a turbine nominal condition, thus the

Figure 4.1: Diagram of rotating components in a modern aero-engine

flow can be treated as steady.

In addition to these, there are other issues needing to be further discussed.

4.2.1 The reduced Static Pressure

The centrifugal force term $f_{cen} = -\Omega \times (\Omega \times r)$, which occurs in the momentum equation can be written as $-\Omega \times (\Omega \times r) = \Omega^2 r e_r$, where e_r is the unit vector in radial direction. And it can be further written in gradient form $\nabla(\frac{\Omega^2 r^2}{2})$, where r represents the distance from the axis of rotation.

This term, normally, is absorbed by the static pressure to form the so called reduced static pressure [100]

$$p_{red} = p - \frac{\Omega^2 r^2}{2} \quad (4.8)$$

The use of the reduced static pressure in rigid rotation with incompressible flow is similar to the procedure of subtracting out the hydrostatic pressure to eliminate the non-dynamical effects

of gravitational forces in a uniform fluid. For an orthogonal rotating channel case, it is useful to work in terms of reduced pressure since it is gradients in reduced static pressure that cause accelerations in the relative system.

4.2.2 The Boussinesq Approximation

Gas turbine rotor blades are cooled by internal flows, and the study of the heat transfer process is very important from a technological point of view. If the Mach number is small and the difference between the wall temperature, T_w , and the inlet fluid temperature, T_0 , is small, and the spatial body force variations can be neglected, the density variations are small, $\Delta\rho/\rho_0 \ll 1$. The Boussinesq approximation can be used to take into account buoyancy effects in the flow and the wall heat transfer in a qualitative fashion. Basically the Boussinesq approximation states that the density variation is only important in the momentum equation, and can be neglected in the rest of them [101].

If express the density as,

$$\rho = \rho_0 + \Delta\rho \quad (4.9)$$

Then the continuity equation, $\nabla \cdot (\rho \mathbf{w}) = 0$ reduces to $\rho_0 \nabla \cdot \mathbf{w} = 0$. This implies that the difference in inertia is negligible but body force term is sufficiently strong to make the specific weight appreciably different between the two fluids with different densities. It allows us to approximate flows with small density variations as incompressible, whilst retaining the leading-order effects due to the density variations.

In orthogonal rotating channels, the centrifugal term is much larger than the Coriolis term in radial direction, therefore only the density variation in centrifugal term is considered, known as centrifugal buoyancy or rotating buoyancy. The temperature and dependent density, have been replaced by a constant density ρ_0 except in the centrifugal force as,

$$\rho \Omega^2 r e_r = [\rho_0 + (\rho - \rho_0)] \Omega^2 r e_r = \rho_0 \nabla \left(\frac{\Omega^2 r^2}{2} \right) + (\rho - \rho_0) \Omega^2 r e_r \quad (4.10)$$

the first part of the equation can be absorbed into the static pressure as the hydrostatic balance, as discussed in previous subsection, and the remaining term is further written in following form:

$$(\rho - \rho_0)\Omega^2 r e_r = -\rho_0 \beta (T - T_0) \Omega^2 r e_r = -\rho_0 \left(\frac{T - T_0}{T_0} \right) \Omega^2 r e_r \quad (4.11)$$

where β is the coefficient of thermal expansion and for ideal gases, $\beta = 1/T_0$. Again, it has to be kept in mind that this is only valid when the temperature difference is small and, therefore, density variations are small. The advantage of the approximation arises because it deals with non-isothermal flow while maintaining the incompressible assumption.

4.2.3 Non-dimensional Form

Consider the assumptions discussed above by using the expression of the reduced pressure (Equation 4.8) and the Boussinesq approximation (Equation 4.11), we obtain the simplified set of governing equations for internal channel flow:

$$\begin{aligned} \nabla \cdot w &= 0 \\ \rho_0 w \nabla \cdot w &= - \underbrace{\nabla p_{red}}_{\text{reduced pressure}} - \underbrace{\left(\frac{T - T_0}{T_0} \right) \rho_0 \Omega^2 r e_r}_{\text{rotating buoyancy}} - \underbrace{2\rho_0 \Omega \times w}_{\text{coriolis}} + \underbrace{\mu \nabla^2 w}_{\text{viscous}} \\ \rho_0 c_p w \cdot \nabla T &= k \nabla^2 T + \underbrace{\mu \Phi}_{\text{dissipation}} \end{aligned} \quad (4.12)$$

In internal cooling problems, the hydraulic diameter of the channel D_h is routinely used as the reference length. Defining u_0 as the mean inlet velocity, ρ_0 , T_0 the inlet density and bulk temperature, r_m is the mean radial of rotation, following non-dimensional terms can be introduced:

$$\begin{aligned} \tilde{x} &= \frac{x}{D_h}, & \tilde{w} &= \frac{w}{u_0}, & \tilde{T} &= \frac{T - T_0}{T_w - T_0} \\ \tilde{r} &= \frac{r}{r_m}, & \tilde{p} &= \frac{p}{\rho_0 u_0^2}, & \tilde{\Phi} &= \frac{\mu \Phi}{u_0} \end{aligned}$$

Put these non-dimensional parameters into Equation 4.12, we obtain the final non-dimensional

form:

$$\begin{aligned}
 \nabla \cdot \tilde{w} &= 0 \\
 \tilde{w} \nabla \cdot \tilde{w} &= -\nabla \tilde{p}_{red} - Bo \tilde{T} \tilde{r} e_r - 2Ro e_x \times \tilde{w} + \frac{1}{Re} \nabla^2 \tilde{w} \\
 \tilde{w} \cdot \nabla \tilde{T} &= \frac{1}{RePr} \nabla^2 \tilde{T} + \frac{Ec}{Re} \tilde{\Phi}
 \end{aligned} \tag{4.13}$$

After obtaining the above set of non-dimensional equations, we see clearly that several non-dimensional numbers have been involved: Reynolds number $Re = \frac{\rho u_0 D_h}{\mu}$, compares the relative importance of the inertia terms to the viscous terms of the fluid; Prandlt number $Pr = \frac{c_p \mu}{k}$ measures the thickness of flow boundary layer compared to thermal boundary layer; Eckert number $Ec = \frac{u_0^2}{c_p (T_w - T_0)}$ accounts for the effect of viscous heating. Additionally, two new non-dimensional numbers appear to characterise the rotating effects.

Rotation number:

$$Ro = \frac{\Omega D_h}{u_0}$$

where it measures the relative importance of the rotational effect (Coriolis effect) compared to the through flow in the cooling channel.

Buoyancy number:

$$Bo = \frac{T_w - T_0}{T_0} Ro^2 \frac{r_m}{D_h}$$

where it accounts for the rotating buoyancy effect, and it can be seen that the effect of rotating buoyancy in channel flows is determined by the combination of density ratio (temperature difference), the Coriolis effect and rotating radius. Furthermore, we can also transform the expression in another form $Bo = \frac{Gr}{Re^2}$, where $Gr = \frac{T_w - T_0}{T_0} \frac{\Omega^2 r_m D_h^3}{u^2}$ is the Grashof number, which denotes the effect of density changes. Therefore, the Buoyancy number also indicates that, for given density ratio and rotational speed, the rotating buoyancy effect is less important if the Reynolds number in rotating channel increases.

In a typical cooling channel of turbine rotor blade, the $Re = O(10^4) \sim O(10^5)$, the Pr and Ec are small, and thus the Rotation number and the Buoyancy number are two important parameters

Figure 4.2: Schematic model of internal cooling channel in turbine rotor blade

to determine the flow.

4.3 Rotational Effects in Internal Cooling Channel

Figure 4.2 shows the schematic model of an internal cooling channel of turbine rotor blade. The main characteristic rotating flow is that: it rotates orthogonally about the engine axis with radially outward flow, and therefore the angular velocity can be simply $\Omega = (\Omega, 0, 0)$. The rotor blade is driven by the main-annulus and therefore always rotates from the pressure side to the suction side of the blade. It is also usual to refer to the two walls of the rotating channel as the pressure side and suction side, or trailing wall and leading wall of rotation. In the following subsections we will describe the main features of the flow inside a rotating channel.

4.3.1 Flow in Two-dimensional Rotating Channels

4.3.1.1 2D channel flow under rapid rotation

Considering a steady isothermal flow in a rotating channel at high Rotation number $Ro \gg 1$, the convective terms, $\tilde{w} \nabla \cdot \tilde{w}$, are negligible compared to the Coriolis force. Assuming that the x-axis is the axis of rotation and neglecting as well the viscous terms, the momentum equation in $y - z$ plane reduces to:

$$\begin{aligned} \frac{\partial \tilde{p}_{red}}{\partial \tilde{y}} &= 2Ro \tilde{w}_z \\ \frac{\partial \tilde{p}_{red}}{\partial \tilde{z}} &= -2Ro \tilde{w}_y \end{aligned} \quad (4.14)$$

The above expression indicates that in a two-dimensional inviscid isothermal flow, the Coriolis force is balanced by the local pressure gradient in the direction perpendicular to the main stream and it may only modify the pressure field without changing the base flow.

If we consider that, the main flow is mainly aligned with the z direction, then $\tilde{w}_z \gg \tilde{w}_y$, and therefore $\tilde{w}_z = w_z/u_0 \approx 1$ and $\frac{\partial \tilde{p}_{red}}{\partial \tilde{z}} \approx 0$, thus we can observed that the reduced static pressure is increasing linearly from suction side (leading surface) to pressure side (trailing surface) of the channel and the variation can be approximately evaluated as,

$$(\tilde{p}_{red})_{ps} - (\tilde{p}_{red})_{ss} \approx 2Ro \quad (4.15)$$

Taylor-Proudman Theorem If we extend the 2D channel to a 3D infinite aspect ratio channel.

By taking the y-derivative and z-derivative of Equation 4.14

$$\frac{\partial \tilde{w}_y}{\partial \tilde{y}} + \frac{\partial \tilde{w}_z}{\partial \tilde{z}} = 0 \quad (4.16)$$

Recall the continuity equation for incompressible flow, and the momentum equation in x-direction, we have,

$$\frac{\partial \tilde{w}_x}{\partial \tilde{x}} = 0, \quad \frac{\partial \tilde{p}_{red}}{\partial \tilde{x}} = 0 \quad (4.17)$$

This expression illustrated that the velocity and pressure are invariant along the rotating axis. If we impose that that w_x is zero somewhere, then it is zero everywhere. This result is known as the Taylor-Proudman Theorem, which stated that, for slow steady inviscid motion of a rapid rotating incompressible fluid, its flow field must be two-dimensional [76, 101].

4.3.1.2 2D rotating channel flow under engine conditions

In the typical cooling channel, the Rotation number is of order unity, Reynolds number is of the order of $Re \sim 10^4 - 10^5$. In this case, we can write the momentum equation in $y - z$ plane as:

$$\begin{aligned} \tilde{w}_y \frac{\partial \tilde{w}_y}{\partial \tilde{y}} + \tilde{w}_z \frac{\partial \tilde{w}_y}{\partial \tilde{z}} &= -\frac{\partial \tilde{p}_{red}}{\partial \tilde{y}} + 2Ro\tilde{w}_z + \frac{1}{Re} \nabla^2 \tilde{w}_y \\ \tilde{w}_y \frac{\partial \tilde{w}_z}{\partial \tilde{y}} + \tilde{w}_z \frac{\partial \tilde{w}_z}{\partial \tilde{z}} &= -\frac{\partial \tilde{p}_{red}}{\partial \tilde{z}} - 2Ro\tilde{w}_y + \frac{1}{Re} \nabla^2 \tilde{w}_z \end{aligned} \quad (4.18)$$

When the Reynolds number is large, we can omit the viscous term in the core flow away from the walls. In this inviscid region, we may further assume that the the flow variation in stream-wise direction can be considered small: $\frac{\partial \tilde{w}_z}{\partial \tilde{z}} \approx 0$, $\frac{\partial \tilde{w}_y}{\partial \tilde{z}} \approx 0$, and $-\frac{\partial \tilde{p}_{red}}{\partial \tilde{z}} \approx 0$. Additionally, from the continuity equation, we have $\frac{\partial \tilde{w}_y}{\partial \tilde{y}} \approx 0$.

The momentum equation can be simplified to,

$$\begin{aligned} \frac{\partial \tilde{p}_{red}}{\partial \tilde{y}} &\approx 2Ro\tilde{w}_z \\ \frac{\partial \tilde{w}_z}{\partial \tilde{y}} &\approx -2Ro \end{aligned} \quad (4.19)$$

Equation 4.19 shows the stream-wise velocity profile has a constant gradient $2Ro$, and the pressure field has the same expression as in the rapid rotation case. If we consider the channel height in a range of $\tilde{y} \in (-1/2 \sim 1/2)$, then the stream-wise velocity and pressure differences can be calculated as:

$$\begin{aligned} \tilde{w}_z &= -2Ro\tilde{y} + 1 \\ (\tilde{p}_{red})_{ps} - (\tilde{p}_{red})_{ss} &= 2Ro\tilde{y} - 2Ro^2\tilde{y}^2 \end{aligned} \quad (4.20)$$

Figure 4.3: Stream-wise flow profile in a 2D channel under rotation

Note that, the above assumptions contains a constant $\tilde{w}_y$ in the inviscid flow region and that needs to be resolved in the boundary layer. Alternatively, we can also consider that, since the flow is inviscid, the absolute flow remains irrotational when enters the rotating channel. Then, the relative vorticity

$$\omega_{rel} = \frac{\partial w_z}{\partial y} = -2\Omega \quad (4.21)$$

giving the same expression,

$$\frac{\partial \tilde{w}_z}{\partial \tilde{y}} = -2Ro \quad (4.22)$$

Figure 4.3 and Figure 4.4 illustrate the numerical results of the stream-wise velocity and reduced pressure profiles across the channel height in a 2D rotating channel. The simulation was conducted using the Mu^2s^2T code at $Re = 12500$, and $L = 9D_h$, where L is the distance from the channel inlet. It can be clearly seen in Figure 4.3 that, as the rotation number increases, the asymmetry behaviour of the stream-wise flow becomes increasingly evident. In the regions away from the walls, the velocity profiles become approximately linear with slope $2Ro$, which is well predicted by the above simplified analysis. Figure 4.4 shows the variation of two-dimensional reduced pressure across the channel, giving also a good match with the predictions. Similar evidence can be found in the DNS results of [102].

Figure 4.4: Stream-wise pressure profile in a 2D channel under rotation

4.3.2 Flow in Three-dimensional Rotating Channels

4.3.2.1 Secondary flows and displacement of stream-wise flow

The above analysis illustrates a number of key features relevant to the flow in rotating channels. In real turbine blades, the aspect ratio of the cooling channels varies from 1 : 4 to 4 : 1 depending on the location inside of the rotor blade, therefore three-dimensional effects has to be considered due to the presence of side walls. Early experimental work of Smirnov and Yurkin [103] and numerical simulations of Kheshgi and Scriven [104] described this secondary flow patterns.

Figure 4.5 compares the secondary flow contours between a stationary and a rotating square channel. It illustrates that, in three-dimensional channel, rotation also creates a cross plane circulation motion. This features can be explained in following ways. The reduced pressure from leading wall of rotation (suction side) to trailing wall (pressure side) under the Coriolis effect of main stream flow. At the side walls where the velocity decreases, the pressure gradient can not be balanced by the local Coriolis force. Therefore, it allows the pressure gradient drives the secondary flow moving from the trailing surface to leading surface along the side wall and moves back at the middle of the channel to close the loop.

In addition to this secondary motion, the Coriolis force of the main stream flow and the pressure

Figure 4.5: Stream-wise velocity field and secondary flow contours in a 3D smooth channel under rotation $Re = 21000$

gradient at the side walls together produces a torque effect whose sense is to cause the stream-wise flow convected towards to the trailing surface [105], which is in the opposite sense to the two-dimensional flow pictured in Figure 4.3. Figure 4.6 illustrates the features of stream-wise flow displacement in a three-dimensional smooth channel under rotation.

Figure 4.6: Stream-wise flow profile in a 3D smooth channel under rotation at $Re = 21000$

4.3.2.2 Ekman layers

As discussed in the previous section, the flow along the side-walls in a 3D rotating channel plays a very important role to characterise the cross plane secondary flow features. Therefore, it is also necessary to discuss the behaviour of this flow. To simplify the problem, we also assume $Ro \gg 1$, to neglect the convective terms. Since the Reynolds number is high, the viscous layers in the vicinity of walls are thin. We can split the flow region into two parts, the core flow, where the viscous flow can be neglected and the viscous flow along side walls.

Outside of boundary layer the governing equations reduce to:

$$\begin{aligned}\frac{\partial \tilde{p}_{red}}{\partial \tilde{x}} &= 0 \\ \frac{\partial \tilde{p}_{red}}{\partial \tilde{y}} &= 2Ro\tilde{w}_z^{Ex} \\ \frac{\partial \tilde{p}_{red}}{\partial \tilde{z}} &= -2Ro\tilde{w}_y^{Ex}\end{aligned}\tag{4.23}$$

where the superscript “Ex” stands for external, referring to the in the core inviscid region outside of the boundary layers. In this inviscid region, recall the Taylor-Proudman theory (Equation 4.17), we have $\frac{\partial \tilde{p}_{red}}{\partial \tilde{x}} = 0$ and $\frac{\partial \tilde{w}_x}{\partial \tilde{x}} = 0$, since the flow is symmetric, we have $\tilde{w}_x = 0$ at the middle of the channel, then it is zero everywhere.

However, inside the boundary layer, since the variation in the direction normal to the wall is much larger than in the other two, $\nabla^2 \tilde{w}_y \simeq \frac{\partial^2 \tilde{w}_y}{\partial \tilde{x}^2}$, and $\nabla^2 \tilde{w}_z \simeq \frac{\partial^2 \tilde{w}_z}{\partial \tilde{x}^2}$, and then,

$$\begin{aligned}0 &= -\frac{\partial \tilde{p}_{red}}{\partial \tilde{y}} + 2Ro\tilde{w}_z + \frac{1}{Re} \frac{\partial^2 \tilde{w}_y}{\partial \tilde{x}^2} \\ 2Ro\tilde{w}_y &= -\frac{\partial \tilde{p}_{red}}{\partial \tilde{z}} + \frac{1}{Re} \frac{\partial^2 \tilde{w}_z}{\partial \tilde{x}^2}\end{aligned}\tag{4.24}$$

Inside of the boundary layer, the pressure is constant in the normal-wise direction, so that the pressure gradient can be obtained from the external flow. Let us now consider the mass balance

of the secondary flow

$$\int_0^{\frac{1}{2}} w_y dx = \int_0^{\delta} w_y^{bl} dx + \int_{\delta}^{\frac{1}{2}} \tilde{w}_y^{Ex} dx \quad (4.25)$$

since the thickness of boundary layer is extremely small compared to external flow region ($\delta \ll 1$), $\tilde{w}_y^e$ will be very small. It is reasonable to set $\tilde{w}_y^e = 0$ and so that $\frac{\partial \tilde{p}_{red}}{\partial \tilde{z}} = 0$ to simplify the problem. The Ekman number is defined as,

$$Ek = \frac{\nu}{\Omega D_h} = \frac{1}{RoRe} \quad (4.26)$$

which represents the ratio between viscous force and the Coriolis force, then we have

$$\begin{aligned} 2(\tilde{w}_z^{Ex} - \tilde{w}_z) &= Ek \frac{\partial^2 \tilde{w}_y}{\partial \tilde{x}^2} \\ 2\tilde{w}_y &= Ek \frac{\partial^2 \tilde{w}_z}{\partial \tilde{z}^2} \end{aligned} \quad (4.27)$$

the above boundary layer has a similar form as the so called Ekman layer which describes the flow balanced between pressure gradient force, Coriolis force, and viscous forces [106]. In our problems, the solution can be obtained analytically with the following boundary conditions

$$\begin{cases} w_y = w_z = 0 & \text{at the wall}(y = 0) \\ w_y = 0 & \text{at } y = 1/2 \end{cases}$$

The final solution has the following form

$$\begin{cases} w_z = w_z^{Ex}(1 - e^{-\eta} \cos \eta) \\ w_y = -w_z^{Ex}(e^{-\eta} \sin \eta) \end{cases} \quad (4.28)$$

where $\eta = x/Ek^{1/2}$, then the length scale, $Ek^{1/2}$ is used to characterise the thickness of the region in which viscous effects are significant. Figure 4.7 plots the profiles of the stream-wise and vertical velocities of Equation 4.28. The negative sign of vertical velocity in y – axis indicates that the flow is from pressure side to the suction side as depicted in Figure 4.5. In addition,

Figure 4.7: Analytical solution of Stream-wise and vertical velocity profiles in Ekman-like layer (Equation 4.28)

the stream-wise velocity and vertical velocity are independent of the y position. Therefore, the thickness of the Ekman layer essentially remains constant. Moreover, the velocity in the wall normal direction, $\tilde{w}_x$, is much smaller than in the other two [105].

In a typical cooling channel, the Rotation number, Ro , is of the order unity, Reynolds number ranges between $Re = 10^4 \sim 10^5$. The convective term can not be neglected, and therefore, the set of equations become non-linear and need to be solved numerically. Although these preceding theoretical studies provide a remarkable insight into the nature of rotating duct flows, non of them was able to determine the magnitude of the secondary flow and as a consequence, predict the flow field accurately since it is greatly affected by the secondary flow convection.

However, even for complex processes, a great deal can be learned from the rigorous use of basic principles. The comparison of theoretical velocity profiles of Ekman layer and numerical results obtained by the in-house RANS solver Mu^2s^2T for an actual channel flows is depicted in Figure 4.8. The numerical results are obtained from a square section 3D channel at the position of $L = 9D_h$. The velocities are taken as averaged value across the channel. Three Rotation numbers were tested $Ro = 0.12, 0.36$, and 1.2 . The corresponding Ekman numbers are $Ek = 4 \times 10^{-3}, 1.33 \times 10^{-3}$ and 4×10^{-4} . The numerical results show that, in three-dimensional

Figure 4.8: Comparison of analytical solution and computational results of Stream-wise and vertical velocity profiles in Ekman-like layer for different Ekman numbers in 3D channels

flow, the characteristic length scale of the Ekman-like layers at side walls is $Ek^{1/2}$, despite the theoretical results are obtained under many assumptions differ from actual flows. It also implies that the physical thickness of the Ekman layer at the side wall reduces with Ekman number increases (recall that $Ek = Re Ro$), and as the Rotation number increases, the flow in the Ekman layer moves asymptotically towards the analytical predictions.

Figure 4.9 shows the reduced pressure field with secondary flow contours in a 3D square section channel. The numerical results are obtained under a constant Rotation number $Ro = 0.12$. It can be seen that as the Ekman number Ek increases, the thickness of the Ekman layer reduces as predicted. Additionally, the reduced pressure field also follows a linear variation, as it was expected.

4.3.3 Coriolis Effects on the Stability of Shear Layer

It has been mentioned that rotating channel flows exhibit substantial alternations in its behaviour under the presence of rotation. The mixing processes and stability of boundary and shear layers are also modified due to rotation.

The mechanism that leads to this alteration can be explained with a simple displaced particle

Figure 4.9: Reduced pressure field and secondary flow contours in a 3D smooth channel under rotation ($Ro = 0.12$)

analysis described by Tritton and Davies [107]. This argument shows the stabilising or destabilising role of Coriolis forces by examining the forces on fluid particles which are displaced from their initial position in the shear layer of a rotating channel. For convenience, let us consider a shear or boundary layer at the trailing wall (pressure side) of the rotating channel. As it is depicted the Figure 4.10. in which the main flow moves along z – axis (radial direction) and the channel rotates about the x – axis.

The Coriolis forces act on the undisplaced particles and are balanced by the local reduced pressure gradient ($\frac{\partial \tilde{p}_{red}}{\partial \tilde{y}} = 2Ro\tilde{w}_z$). Suppose that a small perturbation leads to the blue coloured fluid particle being displaced a small distance $d\tilde{y}$ in the y -direction, and has a new velocity, whose component in z -direction is $\tilde{w}'_{z,1}$. If the disturbance is small, we can consider $\tilde{w}'_{z,1} \approx \tilde{w}_{z,1}$. At its new position of this particle, the Coriolis force can not be balanced by the local pressure gradient (since $\tilde{w}_{z,2} > \tilde{w}_{z,1}$). Thus, the particle will be further pushed away, which means the perturbation has been amplified by the Coriolis force, therefore we call it destabilising effect. It is easy to prove that at the leading surface (suction side) of the channel, the boundary layer is stabilised by the Coriolis force.

The phrase stabilising or destabilising used here means if the stability of a laminar flow is increased (decreased) by external influences, the the corresponding turbulent flow is altered in structure to become in some sense less (more) vigorous [107].

Figure 4.10: Coriolis forces on fluid particles in a rotating channel

To extend the analysis in more general shear flow, we can formulate the disturbing velocity as,

$$\tilde{w}'_{z,1} - \tilde{w}_{z,1} = 2Ro\tilde{w}'_{y,1}d\tilde{t} = 2Rod\tilde{y} \quad (4.29)$$

and

$$\tilde{w}_{z,1} - \tilde{w}_{z,2} = \left(\frac{d\tilde{w}_z}{d\tilde{y}}\right)d\tilde{y} \quad (4.30)$$

from the above analysis, we have already known that the shear layer is stable if $\tilde{w}'_{z,1} > \tilde{w}_{z,2}$, then we obtained that the flow is stabilised if

$$\tilde{\omega}_x + 2Ro > 0 \quad (4.31)$$

where $\tilde{\omega}_x = \frac{\partial \tilde{w}_z}{\partial \tilde{y}} - \frac{\partial \tilde{w}_y}{\partial \tilde{z}}$ is the relative shear vorticity or in general form

$$\tilde{\omega}e_{axis} + 2Ro > 0 \quad (4.32)$$

implement non-dimensional number $S = 2Ro/\tilde{\omega}e_{axis}$, then the flow destabilised $-1 < S < 0$ [108].

The expression indicates that, when the relative vorticity of a shear flow has the opposite sense to the system rotation and large enough, then the Coriolis effects are destabilising the shear layer with reference to non-rotating case. In addition, if the relative vorticity has the same sense that the system rotation, the Coriolis effects stabilise the shear layer.

For instance, if we use the expression of Blasius boundary layer [109] to estimate the stabilisation/destabilisation effect of the boundary layer, we have the shear vorticity

$$\tilde{\omega}_x|_{y=0} \approx -0.332\sqrt{Re_D/\tilde{l}_c} \quad (4.33)$$

where Re_D is the Reynolds number based on the hydraulic diameter and, $\tilde{l}_c = l_c/D_h$ is the relative length of the rotating channel. In typical engine range, we have $\tilde{\omega}_x|_{y=0} \gg 2Ro$, thus we can consider the trailing surface (pressure side) of the channel is always destabilised by rotation.

The mechanism described has implications for boundary layer behaviour in adverse pressure gradients. As mentioned, destabilisation means that momentum interchange is increased (compared to the situation with no rotation) and this increases the resistance of the boundary layer to separation. Stabilisation, with an associated decrease in momentum interchange, has the opposite effect.

4.3.4 Rotating Buoyancy Effects

Internal channels are used to cool the turbine rotor blade, thus there are temperature differences between the channel wall and fluid. The effect of rotating buoyancy is measured by the Buoyancy number $Bo = \frac{T_w - T_0}{T_0} Ro^2 \frac{r_m}{D_h}$ (Equation 4.13). When the temperature difference is significant or the radius of rotation is large enough, a buoyancy effect is created which may further modify the flow field.

Figure 4.11 illustrates the variation of stream-wise flow profile at the middle of a 3D smooth channel under different Rotation and Buoyancy numbers. The results are obtained at $Re = 21000$ at three different conditions: stationary, rotation without heating, and rotating with large buoyancy numbers. It can be seen that the stream-wise flow profiles are significantly different from case to case. In the stationary channel, the flow behaves like typical developing turbulent channel flow, and the flow is symmetric. When rotation is present, as described in the previous subsection, the Coriolis force drives the flow towards trailing wall, and therefore the velocity

Figure 4.11: The rotating buoyancy effect on the stream-wise flow profile in a 3D smooth channel

reduces at the leading wall and increases at trailing wall of the channel.

When buoyancy effects are large, as shown in Equation 4.13, the buoyancy force acts in the opposite direction of the stream-wise flow. Due to the fact that at the leading wall where the velocity is low, the heat can not be effectively convected away from the leading wall, then the nearby fluid temperature is higher than that at the trailing wall, therefore reinforcing the buoyancy effect. As a result, the fluid at the leading wall is slowed down further displacing the flow towards trailing wall (see Figure 4.11 bottom), then the flow at the trailing wall is accelerated to maintain the mass balance. This phenomenon further enlarges the differences between the two walls, and even flow reversal can appear at the leading wall.

4.3.5 Heat Transfer in 3D Rotating Channels

Since rotation modifies the flow and stability of shear layers, it can be anticipated that rotation will also impact the heat transfer performance. The Pioneer work of Wagner et al. [110, 111] showed that rotation enhances the heat transfer on the trailing wall and reduces it on the leading wall of a rotating channel. Because of rotational effects destabilise the boundary layer at the

Figure 4.12: Rotational effect on heat transfer performance for a 3D rotating channel as a function of Rotation numbers

trailing wall, leading to a higher mixing process which enhances the heat exchange. Additionally, the secondary flow provokes the motion of cool fluid at the middle of channel towards the trailing wall of the channel (Figure 4.3 and Figure 4.6). The displacement of fluid creates an impingement effect onto the trailing wall, increasing the momentum of the fluid in this region. Therefore, a higher heat transfer rate is obtained on the trailing wall with respect the stationary case, whereas the opposite is true at the leading walls, where the heat transfer rate is reduced.

Figure 4.12 illustrates the rotational effect on heat transfer obtained by our numerical results and the experimental work of Wagner et al. [112]. The heat transfer data are obtained at $L = 9D_h$ of the channel inlet and normalised with the heat transfer in a stationary channel. It can be seen that, the heat transfer level increases monotonically with the Rotation number. The heat transfer level can be up to three times higher than in a stationary channel. On contrary, the heat transfer at the leading wall reduces as Rotation number increases, however, below a certain Rotation number the heat transfer remains constant. This is because the heat transfer level is so low that becomes insensitive to the fluid field.

Figure 4.13: 3D ribbed channel model

4.4 Flow and Heat Transfer in 3D Ribbed Channels

In order to enhance the heat transfer performance in internal cooling channels, ribs are often used as turbulent promoting devices. The advantage of using rib turbulators is that the rib promoted separation - reattaching flow increases the mixing between the wall boundary layer and the mainstream, enhancing the heat transfer level at the surface. Meanwhile, ribs only disturb the near wall regions, minimising the pressure drop penalty. Thus, it is also important to understand rotational effects in ribbed channels. In the following lines, the fundamental flow and heat transfer behaviour of rib roughened channels is investigated using numerical simulations for well established canonical ribbed channel configurations using the same configuration as that used recently at the von Karman Institute for Fluid Dynamics to obtain new experimental data [113, 114, 115, 116].

The geometry of the ribbed channel is illustrated in Figure 4.13, the channel is one-side ribbed, and the aspect ratio of the channel is $AR = 0.9$, the relative rib height $e/D_h = 0.1$ and the spacing of the rib $p/e = 10$. The channel can rotate in two directions so that the ribbed surface can behave either as trailing wall or leading wall independently. The Reynolds number of inlet flow is $Re = 15,000$, and the temperature differences of the ribbed wall and fluid are small to minimise the buoyancy effects.

4.4.1 Rotational Effects on the Inter-rib Flow

Figure 4.14 shows the streamlines and stream-wise velocity contours of inter-rib flow under different rotating conditions. The flow is taken in the region between 6th and 7th ribs, so that the flow can be considered as fully developed.

The main characteristics of the inter-rib flow is that, an acceleration is observed on the top of the rib due to the blockage. Then a sudden expansion and a low momentum recirculation behind the rib appears. The height of the separation bubble reduces downstream and reattaches at certain point. However, there are significant differences in the size of recirculation bubble. When the ribbed wall rotates as the trailing side, the size of the recirculation bubble reduces compared to that in a stationary case. Due to the destabilisation effect on the trailing wall shear layer induced by the rotation, the exchange of fluid from the separation bubble and main fluid is enhanced. However, when the ribbed wall is placed at the leading wall, the shear layer generated by the ribs is stabilised by the rotation and the entrainment of fluid from the separation bubble to main stream is strongly reduced, resulting in an enlarged recirculation area. Similar behaviours were reported in the experimental work of Mayo et al. [115], although their reattachment lengths are in general shorter than in the numerical simulations. This is due to the fact that most of the one and two equation turbulence models used in RANS simulations have not been designed to account for separated flows. therefore they are not able to fully capture the turbulent fluctuations in the separation region.

4.4.2 Secondary Flow Contours

The secondary contours and pressure field at three rotating conditions are illustrated in Figure 4.15. A growth of the shear layer generated by the rib can be observed in the stationary case. In both rotating cases, the characteristic secondary flow patterns due to rotating can be clearly seen. The static pressure increases almost linearly from the leading side to the trailing side of the channel; and there is a secondary flow moving from the leading side to the trailing side in the middle of the channel, and back to the leading side along the side walls. The main differences with

Figure 4.14: Streamline and stream-wise velocity contours of inter-rib flow under different rotating conditions at $Re = 15000$

Figure 4.15: Secondary flow in one-side ribbed channel under different rotating conditions at $Re = 15000$

Figure 4.16: Heat transfer performance of one-side ribbed channel under different rotating conditions at $Re = 15000$

respect the smooth channel is the interaction between the rib and the rotating flows. When the rib side is the trailing wall, an additional pair of counter-rotating vortexes appears at the middle of the channel in the vicinity of the trailing walls. Mayo et al. [115] also presumes the existence of this vortexes indirectly, despite it is not explicitly shown due to the inherent limitations and difficulties associated with flow measurements in a rotating environment.

In the rib leading rotation case, this pair of vortexes does not appear behind rib because of the lower pressure at the leading wall. However, it can be clearly seen that the shear layer generated by ribs are stabilised by rotation.

4.4.3 Heat Transfer Performance

The heat transfer behaviour at the region between 6th and 7th ribs is shown in Figure 4.16. The heat transfer performance is presented as the Nusselt number normalized with the Nusselt

Figure 4.17: Comparison of sectional averaged Nu/Nu_0 between numerical simulation and experimental data as a function of Rotation numbers

number obtained from the Dittus-Boelter correlation, Nu_0 [117]. This correlation is used for fully developed turbulent flow through a smooth stationary pipe. Therefore, the Nu/Nu_0 ratio is given as:

$$\frac{Nu}{Nu_0} = \left(\frac{h \cdot D_h}{k} \right) \frac{1}{0.023 Re^{0.8} Pr^{0.4}} \quad (4.34)$$

Figure 4.16 (a) illustrates the comparison of span-wise averaged Nusselt number between our numerical data and the experimental work of Mayo et al. [116] for the stationary case. It shows that our RANS solution provides reasonable agreement and similar trend to the experiment. Analogously as it was concluded in smooth channels, the heat transfer level is enhanced when the ribbed wall is located in the trailing surface and reduced when it is in the leading surface, with respect to that of the stationary case.

In order to investigate the overall rotational effect on the heat transfer enhancement, the sectional averaged Nu/Nu_0 has been presented as a function of Rotation number in Figure 4.17. It shows that, the numerical results match well with the experimental data, and present, approximately a linear dependency with respect to the Rotation number. The overall behaviour that the heat transfer is enhanced at the trailing wall and reduced at the leading walls is the same as in a smooth channel (see Figure 4.12).

4.5 Summary

This chapter presents the fundamentals of rotational effects in the flow and heat transfer of internal cooling channels of turbine rotor blades. Theoretical analysis and a number simulations are used to illustrate the impact of the rotation on the flow and heat transfer. The most important rotational effects can be summarised as:

1. There is a linear increase of the reduced pressure from the leading side to the trailing side of the channel due to the contribution of the Coriolis force. The differences can be roughly estimated as $p_{red}^{ts} - p_{red}^{ls} = 2Ro \rho_0 u_0^2$
2. In 3D channels, the Coriolis force forms the secondary motion that drives the flow from the leading wall towards the trailing wall and moves back to leading wall at the side walls. This secondary motion reduces the flow near the leading wall and displaces the velocity peak towards the trailing wall.
3. The rotation in general destabilises the shear layer at the trailing wall, and stabilises the shear layer at the leading wall.
4. Due to the rotational effects described above, the heat transfer performance is enhanced at the trailing wall and reduced at the the leading wall of the channel.
5. The above characteristics are also seen in ribbed channels. In addition, at the trailing side of the rotating channel, the size of separation bubble behind the rib will be reduced due to the destabilise effect whereas the size of the separation bubble will be enlarged at the leading side.

5 Uneven Wall Heating Effect of an Engine Representative Cooling Channel

Contents

5.1	Introduction	124
5.2	Rotating Channel Model	126
5.3	Experimental Set-up	127
5.4	Numerical Set-up	130
5.5	Results	132
5.6	Summary	149

This chapter presents a numerical study on the impacts of uneven wall heating conditions under different Buoyancy numbers of a rotating channel with one side smooth and one side roughened by 45 degree inclined ribs. Parametric RANS simulations were conducted under two different wall heating conditions: only ribbed wall heated, as in experiment setup, and all walls heated, under three different Buoyancy numbers. Results are compared, when available, with experimental results.

5.1 Introduction

The continuous efforts for improving efficiency and thermal performance in modern aero-engines lead to the use of very high turbine entry temperatures and to minimise the amount of coolant flow consumption. Therefore, highly sophisticated cooling techniques are essential to maintain gas turbine safety and durability under such extreme conditions. In order to increase the heat transfer performance of internal cooling systems, complex internal cooling channels are cast in the blades, enhanced by using artificial roughness elements or turbulence promoters such as ribs, pin-fins or dimples. These elements enhance the heat transfer in the cooling channels by triggering the separation and reattachment of the boundary layer, and creating additional mixing between the hot fluid near the wall and the relatively cooler core flow.

Rib-roughened channels, which are the most widespread turbulence promoting device, have been intensively studied over the passed 40 years. There are many parameters affecting the performance of rib-roughened channels, such as, the channel aspect ratio, which can vary within a wide range, depending on the location of the channels inside the turbine blade [118, 119], the entry geometry effect [120, 121], and rib location inside of the channel [122], etc. All of them can affect heat transfer level in a variety of ways.

In addition to these geometrical parameters, heat transfer in rotating channels is strongly influenced by rotational effects. Coriolis and centrifugal buoyancy forces induce additional secondary flows and vortexes in transverse direction along the channel, resulting in very complex flow-fields. As a result, the interaction of the secondary flow induced by rotation and the rib turbulators results in heat transfer patterns differs from non-rotating channels.

Consequently, a considerable amount of literature work has been devoted to evaluate heat transfer performance in rotating channels combined with various forms of geometrical parameters. The pioneer work of Wagner et al. [110] and many others followings [123, 124, 125, 126, 127, 128] have proved that, in general, rotational effects enhance heat transfer performance on the trailing surface, whereas they reduce it on the leading surface in an orthogonal rotating channel with radially outward flow. However, some exceptions were reported for 45 degrees inclined rib

channels in which the heat transfer level is nearly unaffected by rotation, due to the interaction of span-wise secondary flows induced by Coriolis force and rib itself [129, 130, 131]. Overall reviews of internal cooling problems and recent advances can be found in [132, 133, 134].

Furthermore, uneven wall heating is another issue that may affect heat transfer in practise. In real gas turbines, the pressure and suction side of turbine blades experience significantly different thermal loads, due to the differences in the external boundary layer characteristics at both sides [60]. Therefore, this effect has to be taken into account in design process in order to minimise thermal stress. In critical cases, even asymmetric ribs may have to be considered to avoid over-cooling on one side. In addition, this effect is also seen in experiment setups. In some experimental studies on internal cooling problems, it is common to heat only one wall of the channel, especially when optical access is involved in heat transfer measurements (either by using Thermochromic Liquid Crystals [135, 116, 136] or Infrared thermography techniques [137] or fluid measurements [138, 114]). An in-depth discussion on this kind of set-up can be found in Coletti et al. [114].

However, it appears that just a few authors have researched the effect of heating one or more walls at different levels in rotating channels [25-28]. It was concluded that the actual impact of uneven heating conditions on the heat transfer level was significant, especially in the first channel of a U-bend geometry with radially outward flow. They presumed that the impact was due to the Coriolis-induced secondary flows, which altered the local coolant temperature by carrying hotter or cooler fluid towards the leading or trailing wall. However, because of the difficulties associated with simultaneous flow and heat transfer measurements at rotating conditions, particularly for inclined rib configurations, it is still unclear if the changes in the heat transfer are also due to the alteration of the main flow-field. Therefore, it is necessary to systematically study this complex flow and heat transfer behaviour with the help of numerical methods.

With the rapid growth of computing power, CFD has significantly expanded its footprint in internal cooling channel analysis to explore details which are difficult to obtain in experiments. RANS has provided its capability to predict reasonable heat transfer performance and capture

the dominant flow features within acceptable turnaround times [29-33], in spite of its accuracy heavily relying on turbulence modellings [148]. LES has also been widely used to investigate complex flows in cooling channels with high fidelity [149, 150] and many studies have obtained good agreement in flow and heat transfer predictions against experimental data [151, 152]. However, its use in practical engineering problems is limited by the relatively large computational cost compared to RANS. Full scale model simulations, taking into account developing flow, or parametric studies are still not affordable by LES in an industrial environment.

The objective of present chapter is to numerically investigate the impact of uneven wall heating conditions on the internal flow and heat transfer performance of a 45° rib roughened rotating channel using RANS simulations. Our previous experimental and numerical studies [137] for the ribbed wall heated case have shown that rotation reduces heat transfer performance on the trailing wall, which is contrary to most of the existing experiences. Since most of available literature addressing rotating channel with 45 degree inclined ribs consider that all walls are heated, the authors suspect that the uneven wall heating may be one reason of this uncommon behaviour. Therefore, numerical simulations were used to investigate the sensitivity of the heat transfer rate to different heating conditions for different Buoyancy numbers, and draw some conclusions.

5.2 Rotating Channel Model

The testing geometry was designed by SNECMA based on a real HPT rotor blade two-pass internal cooling channel for an experimental campaign. It consists of two rectangular channels connected by a 180 degree sharp turn. The overall dimensions of the test channel can be found in Figure 5.1. The total length of the channel is 350 mm, and the height 70 mm. The width of the upstream and downstream passages is 31 mm and 38 mm, respectively. The gap between the two channels is 7 mm and the distance between the divider wall and outer end wall is 30 mm. The outward upstream passage has an aspect ratio of $AR = 1 : 2.25$ and after the turn, the inward downstream passage has an aspect ratio $AR = 1 : 1.85$. One side of the two channels is

Figure 5.1: Cooling channel sketch

equipped with 45 degree square cross section skewed rib turbulators, with a rib spacing $P = 7e$ and a blockage ratio $e/d_h = 0.116$, where the hydraulic diameter is referred to the upstream passage. In total 14 ribs are arranged in parallel, while the other side remains smooth in order to ease the optical access in experiment.

5.3 Experimental Set-up

The experimental campaign was conducted at ONERA BATHIRE rig facility, which is used for the testing of both, stationary and rotating internal cooling systems. The main test section of the experimental mock-up is shown in Figure 5.2.

The assembly provides structural support and ensures the strength of the whole mock-up. It connects to the rig shaft through a flange which enables the air-supply and the routing of thermocouples and electrical supply for the heating system. The frame is fixed to the assembly. It clamps the lateral windows and the separation wall between the two passages. In front of the measurement wall, there is a window made up of synthetic sapphire, that is transparent in the infrared range, to enable an optical access for the infrared camera.

The measurement wall is fixed to the frame with no contact with the assembly in order to prevent backward thermal conduction losses. The measurement wall consists of three layered components. A stainless steel plate is used whose surface is in contact with the cooling flow stream. This face is black colour painted in order to reach an emissivity close to 1. It is equipped

Figure 5.2: Main test section in experimental set-up

with 5 thermocouples for temperature calibration and measurement. A thin copper plate, on which the stainless steel plate is brazed, is used to uniformize the heating of the stainless steel plate. At last, the heating film is glued to the other side of the copper plate. The measurement plate features a cut-out in which the separating wall is fitted. This cutting prevents lateral conduction between the two branches.

Infrared thermography measurements are made when the mock-up has reached its thermal equilibrium. Monitoring and data acquisition systems enable to follow in real time the evolution of the thermocouples installed on the measurement plate. On the PC screen, inlet and outlet flow temperatures are also displayed. These graphs allow to estimate when the thermal equilibrium has been reached. The test sequence is: (i) Flow injection at a flow rate corresponding to the target Reynolds number. The flow rate is regulated by the monitoring program. (ii) Heating power adjustment by changing the supply voltage for the heating film. (iii) Waiting for the thermal equilibrium and (iv) Recording from the infrared camera. A synchronization mechanism between the camera and the rotating arm is implemented to trigger properly the recording.

In order to determine the effective heat transferred in the measurement wall, calibration tests were performed in advance with the same geometry but with a smooth channel at each Reynolds

number and rotating condition. Wall temperature and reference temperature were measured at a location where the heat transfer coefficient had been determined by previous studies. Because of the use of a highly conductive copper plate next to the measurement wall, heat flux of the measurement wall is considered spatially uniform in the tests. Then, the net heat Q_{net} and effective heat factor, α , can be calculated in each calibration test as,

$$Q_{net,c} = Ah_c(T_{w,c} - T_{ref,c}) = \alpha Q \quad (5.1)$$

where A is the area of the measurement wall, and Q is the total heating power. Three heating powers: 100W, 150W and 200W are used to smooth the error of a single test. The obtained α values are then used for the ribbed cases assuming that the losses remain identical under the same Reynolds number and rotating conditions for the same channel geometry. The evaluation of heat transfer coefficient is done with the effective heating power αQ and wall temperature obtained by IR measurements as,

$$\begin{cases} h = \frac{1}{3} \sum_{i=1}^3 h_i & , i \in [1, 2, 3] \\ h_i = \frac{\alpha_j Q_j / A - \alpha_k Q_k / A}{(T_{w,j} - T_{ref,j}) - (T_{w,k} - T_{ref,k})} & , \forall i \neq j \neq k \\ T_{ref,i} = T_{in} + s \cdot \frac{\alpha_i Q_i}{C_p \dot{m}} \end{cases} \quad (5.2)$$

where C_p is the specific heat capacity, $\dot{m}$ the mass-flow rate, and $s \in [0, 1]$ stands for the curvilinear coordinate of the current location along the centre line of the channel. The reference temperature at any location of the test section is calculated based on the conservation of mean energy.

Consider the effective resolution of the infrared camera, and the potential non-uniformity of actual heat flux into fluid, for providing more reliable data, the final experimental results are provided as section averaged Nusselt number in the area comprised between two ribs, normalized with the Nusselt number obtained from the Dittus-Boelter correlation, Nu_0 [117]. This correlation is used for fully developed turbulent flow through a smooth stationary pipe. There-

fore, the Nu/Nu_0 ratio is given as,

$$\frac{Nu}{Nu_0} = \left(\frac{h \cdot D_h}{k} \right) \frac{1}{0.023 Re^{0.8} Pr^{0.4}} \quad (5.3)$$

where heat transfer coefficient h is obtained by Equation 5.2 and D_h is the hydraulic diameter of inlet passage.

5.4 Numerical Set-up

Entry shape can have a significant impact on the flow-field and heat transfer performance of rotating channels [120, 128]. Therefore, the computational domain was extended to include the supply system in order to account for inlet entry effects. Figure 5.3 shows the computational domain and mesh used in the numerical simulations. The inlet/outlet supply system has a circular pipe shape that transitions to a C-shaped pipe near the joint with the transition pipe to create an engine-like entry flow. The model is discretised by means of unstructured hybrid grids using an in-house mesh generator. The grid consists of a near-wall prismatic layer region, while the rest of the domain is discretised using isotropic tetrahedral elements. In total, 18 prismatic layers are constructed with $y^+ < 1$ for the tested Reynolds number. The total grid size is 13.5 million cells. A coarser mesh with 8.1 million nodes was used to perform a mesh sensitivity check, and the results is shown in Tab. Table 5.1. The change in the segment averaged heat transfer coefficient was less than 2%. Hence, the solution obtained in the current study is considered grid independent.

Steady-state simulations have been conducted using an in-house RANS code Mu^2s^2T [91]. The code solves the 3D RANS equations using an edge-based data structure to spatially discretise the equations using a MUSCL scheme. 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. Low Mach preconditioning is used to enhance the convergence and accuracy for low Mach number flows. The code is fully parallelized using MPI and executed on GPUs to improve computational efficiency.

Nodes per segment (total cells)	y+	Avg. htc
66×45 (13.5M)	$<1, <3$	0%, -0.9%
44×30 (8.1M)	$<1, <3$	1.1%, -1.6%

Table 5.1: Mesh sensitivity study

Heating condition	Ro	Bo
(a) ribbed wall heated	$0, \pm 0.3$	0.017, 0.17 (Exp.) , 0.34
(b) all walls heated	$0, \pm 0.3$	0.017, 0.17, 0.34

Table 5.2: CFD test matrix

The turbulence effects were accounted for using the $k - \omega$ SST turbulence model [153] with an inlet turbulence intensity of 5%. The inlet/outlet boundary conditions have been obtained from the experimental measurements. Total temperature and mass flow rate are provided at the inlet, and static pressure is set at outlet. This numerical study was performed at Reynolds number $Re=15,000$ under three rotating conditions: stationary ($Ro = 0$); forward rotation direction ($Ro = 0.3$), where the ribbed surface behaves as the trailing wall (pressure side) of a rotor blade; and backward rotation direction ($Ro = -0.3$), where the ribbed surface behaves as the leading wall (suction side).

In order to investigate the effect of wall heating conditions, two wall boundary conditions were tested: (a) only ribbed wall heated; (b) all walls heated. These two heating conditions are simulated under three different Buoyancy numbers calculated at a fixed Rotation number ($Ro = 0.3$) in order to separate the buoyancy effect: (i) $Bo = 0.017$, where the buoyancy effect is small; (ii) $Bo = 0.17$, the nominal condition in the experiment; and (iii) $Bo = 0.34$.

The Buoyancy number is defined as:

$$Bo = Ro^2 \frac{(T_w - T_{in})}{T_{in}} \frac{r_{mean}}{d_h} \quad (5.4)$$

where r_{mean} is the mean radial distance of the cooling channel to the rotating axis, and d_h stands for the hydraulic diameter of the first channel. The supply system is set as adiabatic in all cases. In each case, a constant temperature distribution is assigned to the heated wall/s to control the mean Buoyancy number. For each rotating case, a non-rotating simulation with the same

Figure 5.3: Hybrid mesh of computational domain

wall temperature is performed as a reference. The overall boundary conditions can be found in Figure 5.4, and the test matrix is shown in Tab. Table 5.2. In total, 18 cases were calculated. The flow domain was partitioned in 8 domains and run in an in-house GPUs cluster powered by Nvidia GeForce GTX 780 Ti. Every single solution was obtained in around 4-5 hours starting from an uniform initial solution.

5.5 Results

For the sake of clarity, hereby the author only considers the analysis of the wall heating effect in the first pass, where the effects of the Coriolis and buoyancy forces are more conspicuous. For the flow domain investigation, we take the segment no. -3 of the first pass, since it is hardly influenced by the bend and the entrance. The region of interest and segment notation are

Figure 5.4: Numerical model and boundary conditions

Figure 5.5: Segment index notation

illustrated in Figure 5.5.

5.5.1 Segment Averaged Heat Transfer Validation

Before performing the parametric study, the numerical results were validated using the experimental data at the nominal condition. The comparison of the normalised segment averaged Nusselt number based on the smooth area between ribs is presented. The convective heat transfer coefficient in the simulations is calculated as,

$$h = q / (T_w - T_{ref}) \quad (5.5)$$

whereas the local reference temperature, T_{ref} , is calculated based on the conservation of mean energy, in the same way that it was performed in the post-processing of the experimental data:

$$T_{ref,s} = T_{in} + s \cdot \frac{Q}{C_p \cdot \dot{m}} \quad (5.6)$$

where T_{in} is the temperature at the channel inlet, C_p is specific heat capacity, Q the total heat, $\dot{m}$ the mass-flow rate, and $s \in [0, 1]$ stands for the curvilinear coordinate of current location along the centre line of the channel. By convenience, to isolate the impact of Reynolds number, the Nusselt number is divided with the Dittus-Boelter correlation [117].

$$\frac{Nu}{Nu_0} = \left(\frac{h \cdot d_h}{k_f} \right) \frac{1}{0.023 Re^{0.8} Pr^{0.4}} \quad (5.7)$$

Figure 5.6 shows the comparison between the experimental and numerical results for the ribbed wall heated case at $Bo = 0.17$ in the first passage. It can be seen that, the segment averaged heat transfer data in the non-rotating and rib trailing rotating cases are well predicted. For the rib leading rotating case, the numerical results over-predict the negative effect of rotation, underestimating the heat transfer in overall by about 30%-35% compared to the experimental data. Because rotation stabilises the flow at the leading wall and enlarges the separation zone, especially when the wall is heated [114]. The flow in this case is fully separated causing difficulties to the turbulence model to accurately predicts the heat transfer rate, since the commonly used turbulence models are not designed to account for the turbulent fluctuations generated inside a recirculation bubble. A turbulence model behaves as a laminar case inside a large separation bubble and the computed heat transfer in this region is correspondingly lower since the physical existing fluctuations are disregarded in practise. However the heat transfer decrease due to the separation is qualitatively predicted.

The most interesting result is that, on the contrary than in most of the previous works, the heat transfer efficiency drops in the ribbed trailing rotating case, compared to stationary case. This behaviour is consistently observed, both in the experiments and in the numerical simulations. Moreover the magnitude of the drop in the rib trailing case, is large, being more than 25%. This

Figure 5.6: Comparison of segment averaged Nu/Nu_0 of the first passage at nominal condition ($Re = 15000$, $Ro = 0.3$ and $Bo = 0.17$), with the ribbed wall heated

variation is much larger than the uncertainties that could be reasonably associated to either the experimental or numerical techniques used in here, and therefore, credibility is given to these results.

5.5.2 Uneven Wall Heating Effects on Heat Transfer Distribution

Figure 5.7 shows the contour plots of the normalised Nusselt number distributions obtained from the numerical simulations for different heating conditions at $Re = 15000$. In the absence of rotation, it is shown that the two heating conditions provide the same heat transfer level and pattern, which is a clear indication that the sensitivity of the flow-field to the different wall heating conditions is very low.

For the ribbed trailing rotating test, the all wall heated case has a significantly higher heat transfer rate than the ribbed wall heated case. The overall patterns are similar, but the peak regions in all walls heated case are enlarged and displaced slightly upstream. This suggests that the main flow features in the vicinity of the ribbed wall have not substantially changed. However, the flow reattachment behaviour and the absolute level may have been modified and should be further investigated.

Figure 5.7: Numerical results of Nu/Nu_0 distribution for two different wall heating conditions at nominal condition ($Re = 15000$, $Ro = 0.3$ and $Bo = 0.17$)

Figure 5.8: Evolution of overall Nu/Nu_0 for uneven heat conditions as a function of the Buoyancy and Rotation numbers

Geo. Parameter	θ	AR	p/e	e/d_h	no. heated walls
Present study	45°	1 : 2.25	7	0.116	single and all
Tanda [136]	45°	4 : 1	6.66	0.09	single
Jenkins et al. [154]	45°	1 : 2	10	0.1	all
Mayo et al. [116]	90°	0.9 : 1	10	0.1	single

Table 5.3: Geometry parameters of one-wall ribbed channels

In the rib leading rotating test, very low heat transfer rates are observed in both cases. Therefore, it can be conjectured that large recirculation bubbles are dominating the inter-rib regions in both cases. However it is not possible to deduce the detailed flow behaviour from this heat transfer pattern since it is quite low.

5.5.3 Evolution of Overall Heat Transfer Performance

Figure 5.8 illustrates the evolution of overall heat transfer performance of the previously described case for two heating conditions as a function of the Buoyancy and Rotation numbers. Results obtained by other researchers at the same Reynolds number are included as a reference. It is worth noting that, first the amount of experimental evidences for one wall ribbed channels is much less than for two wall ribbed channels, and second this is a factor that needs to be taken into account, since in general, the latter have higher heat transfer level than the former [136]. A summary of the one wall ribbed channel studies used for comparison with the present study is shown in Tab. Table 5.3, geometrical differences are highlighted.

It is shown that, for the non-rotating cases, the sensitivity of the overall heat transfer performance to changes in the heating conditions is very low. Numerical simulations match well with our own experiments and the data obtained by Jenkins et al. [154]. Tanda [136] reports a heat transfer rate around 8% lower than our own results, but it is within a reasonable range since the aspect ratios of both tests are different.

In the ribbed trailing rotating tests, we have a good agreement with our own experimental data, however large differences with Mayo et al. [116] due to the uncommon behaviour in heat transfer of this case that we are reporting here. It is important to recall here that Mayo et al. [116] tested a cooling passage with straight ribs, whose flow is significantly different to that

Figure 5.9: Effect of uneven heating under rotation as a function of the Buoyancy number Solid symbols: all walls heated; hollow symbols: rib wall heated

encountered in inclined ribs, as in our case. The numerical results show that the heat transfer performance is very close for the two heating conditions when buoyancy effect is negligible. The heat transfer performance for the all-wall heated case increases monotonically with the Buoyancy number, which is a widely admitted trend [132]. However, the heat transfer level is significantly higher than for the single ribbed wall heated case, and the difference between both cases increases with the Buoyancy number, because for the latter the heat transfer rate slightly drops with the Buoyancy number.

This trend can be more clearly seen in Figure 5.9, in which the Nusselt number is normalized by the Nusselt number of the lowest buoyancy case in each individual setup to isolate flow and

Test Parameter	θ	Ro	Re	no. heated walls
Present study	45°	0.3	15k	single and all
Johnson et al. [123]	45°	0.25	25k	all
Mayo et al. [116]	90°	$0.38 \sim 0.84$	15k	single
Abdel-Wahab & Tafti [155]	90°	0.36	20k	all

Table 5.4: Test parameters of rotating channels

geometrical differences (Tab. Table 5.4). The solid symbols represent all-wall heated cases, whereas the hollow symbols mark single ribbed wall heated cases. Mayo et al. [116] reported that they have good agreement for low buoyancy cases with other authors except with Abdel-Wahab & Tafti [155] at high buoyancy cases. They conjectured that this could be due to the boundary conditions. According to the current study, the uneven heating condition may also be a plausible explanation. Figure 5.9(a) clearly shows that there are two different trends of the heat transfer rate with the Buoyancy number, depending on whether all the wall or a single wall is heated. This behaviour can be noticed both in the experiments and different set of experimental data and therefore this statement deserves a lot of credit.

In our experiment we have found that, on the contrary to existing body of knowledge, the heat transfer rate for the ribbed trailing rotating case is significantly lower than for the non-rotating case. In addition to geometrical differences, we associate this fact to the uneven heating conditions since it can be seen that when the simulations are performed heating all the walls, instead of heating only the ribbed wall, as in the experiment, the trend is to increase the heat transfer and reduce the difference with the stationary case (see Figure 5.8). For high Buoyancy numbers ($Bo > 0.34$), the all-wall heated cases behave as it could be expected.

In the ribbed leading rotating tests, Figure 5.8 and Figure 5.9(b) shows that both heating conditions the heat transfer level is reduced as the Buoyancy number is increased. However, the overall heat transfer sensitivity to the heating conditions is not significant(see Figure 5.8). This is because, the heat transfer level is already very low, and the variation of flow field may not show significant impact on heat transfer performance. This case will be further discussed in the analysis of the flow-field.

It is concluded that the trends of our experiments and simulations show reasonable agreement

with the experimental data independently obtained by other researchers, when the differences of channel geometry parameters are taking into account. Therefore, the conclusions about the overall heat transfer evolution with respect to the heating conditions and the Buoyancy number are credible.

5.5.4 Secondary Flow Characteristics

The heat transfer behaviour in rotating channels is strongly dependent on the flow characteristics, therefore, it is compulsory to analyse the complex flow-field generated in the passage to understand the aforementioned heat transfer trends. Figure 5.10 displays the relative velocity magnitude and secondary flow patterns at the channel inlet. The relative velocity is normalised with the mean inlet velocity U_0 . The inlet velocity contours show that the entry geometry formed a counter clock-wise secondary flow at the middle of the channel in the non-rotating case. This inlet induced secondary flow is weakened in the ribbed trailing rotating case, whereas it is preserved in the ribbed leading rotating case due to the interaction with the Coriolis induced secondary flow. It is clearly seen that, under rotation, the inlet velocity profiles are significantly altered with respect to the non-rotating case. However, the entry flow profiles are identical for all heating conditions, since there is no heating in the air supply system. Thus, it will not be shown repeatedly.

Figure 5.11(A) shows a parametric study of the secondary flow pattern at the middle of segment no. -3 (yellow line of Figure 5.5). In the non-rotating tests, the presence of inclined ribs (upper side) induces a high momentum span-wise flow along the rib creating a strong clockwise vortex in the vicinity of the ribbed wall region. Unlike the straight rib channel, in which the heat transfer is mainly dependent on the stream-wise separation-reattaching flow, this span-wise motion strongly affects the heat transfer behaviour by increasing the flow velocity, and bringing cooler fluid to the rib wall [131]. Meanwhile, the counter clockwise vortex, formed at the entry shape, is pushed to the middle part of the channel. It is observed that the flow-field remains unchanged in all the non-rotating cases. This indicates that the heat is effectively convected downstream, and neither wall heating conditions, nor temperature differences can

Figure 5.10: Velocity magnitude and secondary flow patterns at channel inlet

produce significant variations in the main flow-field (Buoyancy numbers refer to the rotating case $Ro = 0.3$).

In the ribbed trailing rotating tests, rotation induces a secondary flow where fluid is transported from the leading surface (bottom) to the trailing surface (top) of the channel. At the side wall, where the fluid is moving slowly, the pressure gradient creates flow with the opposite sense, moving from the upper side to the lower side, to close the vortex. Figure 5.11(B) shows that the secondary flow patterns are essentially the same, because of the span-wise secondary flow is determined by the Coriolis force, and buoyancy force does not have a direct contribution.

However there are significant changes in the velocity field as Buoyancy number increases. When the Buoyancy number is small ($Bo = 0.017$), similarly as in the non-rotating tests, the uneven heating does not alter main flow field significantly. At high Buoyancy numbers ($Bo = 0.17$, and 0.34), the ribbed wall heated and the all walls heated cases differ significantly, since in the latter the velocity profiled is skewed towards the upper ribbed trailing surface. In addition, a reduced velocity appears at the leading surface in the all-wall heated case. This phenomenon is observed to increase with the buoyancy number. The underlying reason is that when all the walls are heated, the leading surface has lower heat transfer, and therefore higher temperature, lower density. As a result, the buoyancy force acts in the opposite direction to the stream-wise

Figure 5.11: Velocity magnitude and secondary flow patterns (arrows) at the middle of segment no. -3 for uneven wall heating conditions under different Buoyancy numbers

flow, the hot fluid near the leading surface is slowed down.

In the ribbed leading rotating tests in Figure 5.11(C), a very low velocity region appears in the vicinity of the ribbed wall (top), suggesting that the heat transfer level is much lower than that in the non-rotating and the ribbed trailing rotating tests. The secondary flow shows similar patterns for different wall heating conditions, moving fluid from the leading side to the trailing side and back to the leading side along the side wall. The presence of the central vortex is due to the interaction of secondary flow induced by the Coriolis force and the entry shape.

At high Buoyancy numbers, the flow in the vicinity of ribbed wall exhibits the opposite behaviour compared to the rib trailing rotating cases due to the change of the rotating direction. The velocity profiles are displaced towards to the trailing surface (bottom). It is worth noting that, in the all walls heated case, the velocity displacement is higher than the ribbed wall heated case. This is possibly because, when all walls are under heating, the secondary flow along the heated side walls transports hot fluid back to the leading surface, further increasing the fluid temperature in that region, and therefore the buoyancy effects. This behaviour may also explain the slight drop in heat transfer level in the all walls heated case.

5.5.5 Inter-Rib Flow Characteristics

The impact of wall heating conditions in the vicinity of ribbed wall can be investigated by analysing the inter-rib flow field. Figure 5.12 and Figure 5.13 depict the stream-wise and span-wise velocities at the inter-rib region for different wall heating conditions at the middle of segment no. -3 (red line in Figure 5.5). The main characteristics of the flow in the non-rotating channel are that, the flow accelerates on the top of the rib due to the blockage in the cross section area, then a sudden expansion and a recirculation bubble with low momentum fluid are created. The height of recirculation bubble is reduced due to the mixing with high momentum flow near the rib region. In addition, in the non-rotating cases, the magnitude of the span-wise velocity induced by the inclined ribs is very high. Figure 5.13 shows that, in the non-rotating cases, the span-wise and stream-wise components of the velocity are of the same order. This span-wise high momentum flow induces additional mixing that in turn fosters a faster re-attachment in

the inter-rib recirculation region, and therefore, high heat transfer can be expected. Comparing the velocity contours, the same conclusions as those in the previous subsection can be drawn, namely that wall heating conditions and temperature differences only have a minor impact on the flow-field in the non-rotating cases.

The ribbed trailing rotating tests are displayed in Figure 5.12(B). When the ribbed wall behaves as a trailing surface, on the top of the already existing rib induced flow of the non-rotating case, we need to superimpose the destabilising effect of rotation, which enhances the entrainment of fluid from the recirculation bubble to the main flow. However, because of the Coriolis induced and rib induced secondary flows are acting in the opposite sense near the rib region (Figure 5.11(B)), this interaction drastically reduces the magnitude of span-wise flow, so does the additional mixing compared to that of the non-rotating case (Figure 5.13), resulting in a larger separation area. Their combined effects lead to the ultimate reason of why the heat transfer rate of this rotating case is lower than in the non-rotating case.

For the lowest buoyancy case, the flow field of the two different heating conditions remains essentially the same, but when buoyancy is increased, the heating conditions alter significantly the flow field. This phenomenon can be explained by the schematic model shown in Figure 5.14. Because of heating, the recirculation region behind the rib has a higher temperature than the core flow, therefore the rotating buoyancy force acts in the opposite direction of the stream-wise flow. Thus, the recirculation area is slightly enlarged in presence of buoyancy. This also explains why the heat transfer slightly drops with the Buoyancy number in the one wall heated cases.

However, when all walls are under heating, due to the fact that the leading surface has a lower heat transfer rate than the trailing surface, the adjacent fluid is hotter and the density lower than that near the trailing surface. As a result, the flow near the leading wall experiences a higher buoyancy force than the trailing wall (Figure 5.14). By the contribution of unbalanced buoyancy forces, the flow velocity near leading surface is slowed down and pushed towards the trailing surface to conserve the mass. This flow motion enhances the mixing with the separation bubble by increasing the momentum of the near wall fluid both, in stream-wise and span-wise directions (Figure 5.13), as well as the Coriolis effect. As a consequence, the size of recirculation bubble

Figure 5.12: Stream-wise velocity contours and streamlines in inter-rib region for different wall heating conditions under different Buoyancy numbers

Figure 5.13: Span-wise velocity contours in inter-rib region for different wall heating conditions under different Buoyancy numbers

in the all-wall heated case is slightly reduced compared to that of the ribbed wall heated case, and a higher heat transfer rate can be expected.

When the ribbed wall behaves as the leading surface, the shear layer generated by the rib is stabilised by the rotation and the entrainment of fluid from the recirculation bubble to the main stream is strongly reduced, resulting in a much larger recirculation area. Analogously to the previous cases, when the Buoyancy number is low, the sensitivity of the flow to the wall heating conditions is low as well. As the Buoyancy number increases, the recirculation bubble is further enlarged both in stream-wise and wall normal directions, and extends even beyond the whole inter-rib region, this phenomenon has also been reported in Coletti et al. [114] in a 90° rib-roughened channel. In addition, in the all walls heated cases, the secondary flow also brings hot fluid from the trailing surface back to the leading surface along the heated side walls, giving rise to a further increase of the local temperature, enlarging the buoyancy effect. Thus, the main stream is further pushed towards to the trailing surface of the channel, resulting in a very low velocity near the rib region. As a consequence, very low momentum fluid both, in the stream-wise and span-wise directions leads to a very low heat transfer rate, as shown in Figure 5.8.

5.5.6 Main Stream Flow

The stream-wise velocity profiles at the middle plane of segment no. -3 along the channel height are presented in Figure 5.15. The channel height is normalised with the height of the rib. It can be seen that due to the inlet velocity profile, and the presence of one side ribbed wall, the stream-wise velocity profiles of non-rotating cases do not correspond to that expected for a fully developed channel flow. However it can be seen in Figure 5.15 (left) that the uneven wall heating has no impact on the stream-wise velocity of the non-rotating cases.

In the ribbed trailing rotating tests, it can be noticed that, when the Buoyancy number is small, surface heating conditions does not produce significantly changes in overall flow-field, but when buoyancy is very large, the strength of the recirculation bubble is enhanced (see the inset in Figure 5.15 (middle)). However, the height of the recirculation bubble is slightly shorter in

Figure 5.14: Schematic model of flow in ribbed trailing wall tests, under Coriolis and buoyancy forces

Figure 5.15: Stream-wise velocity profile along channel height at middle plane of segment -3 for different wall heating conditions under different Buoyancy numbers

the all walls heated cases than in the ribbed wall heated cases, due to the contribution of the displacement of main stream flow.

In addition, when only the ribbed wall is heated, it is observed that the buoyancy effect slightly alters the velocity field in the vicinity of the ribbed wall, and the overall flow field is almost unaffected. Meanwhile, when all the walls are heated at high Buoyancy numbers, the stream-wise velocity near the leading surface (top) is strongly reduced, and even flow reversal can be seen (at $Bo = 0.34$). The fluid core is, on the whole, pushed towards the ribbed trailing wall (bottom).

In the ribbed leading rotating tests, Figure 5.15 (right) depicts that, the height of separation bubble is enlarged by buoyancy. Furthermore, when all the walls are heated the height of separation bubble is further increased, indicating the effect of buoyancy on the leading surface is enlarged, further displacing the fluid core towards the trailing surface. However, since the heat transfer rate is already very low in this very low flow momentum region, the variation of heat transfer rate with the heating conditions is not conspicuous.

5.6 Summary

The impact of the uneven wall heating conditions, under different Buoyancy numbers on the flow-field and heat transfer performance of a rotating channel, with one side smooth and another one roughened by 45 degree inclined ribs has been investigated. RANS simulations were conducted under two different wall heating conditions: (a) ribbed wall heated cases, and (b) all walls heated cases, under three different Buoyancy numbers. Results are compared, when available, with experimental results. The conclusions can be summarised as follows:

1. For non-rotating cases or low Buoyancy number rotating cases, the heat transfer and flow-field of the cooling channels are hardly affected by the heating conditions.
2. For the ribbed wall heated cases at high Buoyancy numbers, the size of the recirculation bubble behind the rib is enlarged reducing the heat transfer level. However, it has less

impact on the bubble size in the ribbed trailing case than in the ribbed leading case due to the flow destabilising/stabilising effect.

3. In the ribbed trailing rotating cases, the heat transfer is significantly higher when all the walls are heated than when just the ribbed wall is heated. The heat transfer in the all walls heated cases increases monotonically with the Buoyancy number, whereas in the ribbed wall heated case is slight reduced. The reason underlying this behaviour is a displacement of core flow towards the trailing surface in all the walls heated cases. Consequently, it brings cooler, high momentum fluid in the vicinity of the ribbed trailing surface. Therefore, it increases heat transfer performance.
4. In the ribbed leading rotating tests, the heat transfer sensitivity to the heating conditions is not conspicuous and for both cases is slightly reduced as the Buoyancy number increases.

According to the results obtained in this work, we conclude that buoyancy plays a predominant role in uneven heating problems, both in heat transfer and flow-field. Thus, when buoyancy effects are relevant, this uneven heating issue has to be considered in the experimental setup and the application of the results in an industrial and more realistic environment.

6 Conclusions and Future Work

This thesis focuses on the internal cooling problems of gas turbine rotor blades in three different aspects: 3D grid generation for complex geometries: the coupled fluid/solid heat transfer method and the rotational effects on flow and heat transfer performance in rotating cooling channels.

Concerning the grid generation, a robust and efficient method is described to generate viscous boundary layer meshes for complex turbine internal cooling geometries in a hybrid fashion. The final mesh is created in a prismatic-tetrahedral hybrid manner for computational fluid dynamics (CFD) calculations. We first give a brief introduction and review of the boundary layer mesh generation methods. Then the implementation of boundary mesh generation method with an enhanced validity checking is described in detail. Finally, a new approach is proposed to improve the efficiency of the baseline algorithm. The accelerated method employs a pre-processing algorithm, called the Outer Layer Boundary (OLB) checking scheme, to identify in advance the cells which can be generated without performing expensive intersection checking operations against other cells of the front. The method substantially reduces the number of intersection checking operations compared to classical advancing layer methods, significantly reducing the mesh generation wall-clock time while maintaining the same level of robustness and quality as the baseline algorithm. For the cases presented here, it is concluded that the new method, named as the “Skyline” method, is between 4 and 10 times faster than the standard one. Moreover it is concluded that the new method is significantly faster as well than other well established hybrid mesh generation systems. Meshes for three different cases of industrial complexity have been presented. This approach has proved to be useful for large size 3D complex configurations

where mesh generation time and interactivity are crucial.

In the study of loosely coupled method, an improved approach is proposed to reduce the computational time of loosely coupled fluid/solid heat transfer methods for steady state heat transfer analysis. Convergence analyses based on a simple one-dimensional model show that, in the Dirichlet-Robin approach, the convergence behaviour is influenced by the physical Biot number and can be controlled by a user specified virtual HTC. The closer the virtual HTC distribution is to the physical one, the faster the convergence rate. A bias error appears when CFD solutions are not fully converged. Thus, the selection of CFD convergence level has to be considered carefully. In this case, fully converged CFD solutions are required during the last few iterations. Using this 1D analysis, a new method based on a dynamic updating process for seeking the optimal virtual HTC is proposed. The new method frees the user, from the selection of a constant virtual heat transfer coefficient that ideally should take into account the local characteristics of the different flow regions. Two realistic 3D configurations have been studied to test the performance of the proposed method. It is shown that the 1D stability criterion can be used to predict the convergence behaviour of 3D complex configurations as well. The dynamic virtual HTC approach provides faster convergence than the baseline one, and, what is more important, it is robust. Further reduction in computational time is obtained using incompletely converged CFD solutions during the first few iterations with a proper selection of the CFD convergence level without corrupting the global coupling scheme and final solution. The use of fully converged CFD solutions requires fewer number of iterations than that of incompletely converged solutions but is computationally more intensive. By using the incompletely converged CFD scheme, the extra cost of running a fully coupled solid-fluid heat transfer problem is estimated in about 35% of the computational cost of the equivalent uncoupled CFD solution.

The study on the rotational effect shows the impact on the flow and heat transfer behaviours in a simple channel and engine representative channels. The analysis has shown that even for complex flow there is a clear understanding of the mechanisms behind it. Despite the flow is strongly influenced by channel geometries and different rib arrangements. The main characteristic of rotation can be retained and used to explain the problem.

The most important rotational effects can be summarised as:

1. There is a linear increase of the reduced pressure from the leading side to the trailing side of the channel due to the contribution of the Coriolis force.
2. In 3D channels, the Coriolis force forms the secondary motion that drives the flow from the leading wall towards the trailing wall and moves back to the leading wall at the side walls. This secondary motion reduces the flow near the leading wall and displaces the velocity peak towards the trailing wall.
3. The rotation in general destabilises the shear layer at the trailing wall, and stabilises the shear layer at the leading wall.
4. Due to the rotational effects described above, the heat transfer performance is enhanced at the trailing wall and reduced at the the leading wall of the channel.
5. The above characteristics are also seen in ribbed channels. In addition, at the trailing side of the rotating channel, the size of separation bubble behind the rib will be reduced due to the destabilizing effect whereas the size of the separation bubble will be enlarged at the leading side.
6. In engine representative configurations, more factors could be involved to influence the heat transfer performance of rotating channel, for instance, the uneven wall heating effect.

Future work

The internal cooling problems of gas turbine rotor blade is difficult for numerical simulation because of its complex flow physics. For instance, the very low momentum flow may result in numeral difficulties for density based CFD solver. In addition, the common used turbulent models did not taken into account the Rotational effect on turbulence structures, and are not designed to simulate impingement or separating flow, which are always encountered in rotating cooling channels. Because of these complexities, the numerical tools are not always reliable for conducting real designs. A possible direction for future work could be the development of new

turbulent model in order to capture rotational effects and the anisotropy behaviour of Reynolds stresses.

Another direction could be the study of the rotating buoyancy effect in real engines. In the last chapter of the thesis, it has been shown that rotating buoyancy could result in uncommon behaviours in heat transfer performance. Furthermore, most of theoretical and experiment studies are designed for small Buoyancy numbers so that the rotational effect can be isolated, however in heavily heated HPT blades, the temperature differences between cooling flow and blade metal could be large and the Boussinesq approximation is no longer valid. Since it is difficult to accurately control such large temperature differences in experiments, theoretical or numerical methods are required to enhance the understanding of the rotating flow under these conditions.

Bibliography

- [1] Tafti, D. K., He, L., and Nagendra, K., 2014. “Large eddy simulation for predicting turbulent heat transfer in gas turbines”. *Philosophical Transactions of the Royal Society of London A: Mathematical, Physical and Engineering Sciences*, **372**(2022), p. 20130322.
- [2] Pirzadeh, S., 1996. “Three-dimensional unstructured viscous grids by the advancing-layers method”. *AIAA journal*, **34**(1), pp. 43–49.
- [3] Aubry, R., and Löhner, R., 2008. “On the most normal normal”. *Communications in Numerical Methods in Engineering*, **24**(12), pp. 1641–1652.
- [4] Haselbach, F., and Parker, R., 2012. “Hot end technology for advanced, low emission large civil aircraft engines”. In 28th International Congress of the Aeronautical Sciences, Vol. 2, p. 3.
- [5] Corral, R., and Fernandez, J., 2001. “Surface mesh generation by means of steiner triangulations”. *AIAA journal*, **39**(1), pp. 176–180.
- [6] Han, J.-C., Dutta, S., and Ekkad, S., 2012. *Gas turbine heat transfer and cooling technology*. CRC Press.
- [7] Dawes, W., 2007. “Turbomachinery computational fluid dynamics: asymptotes and paradigm shifts”. *Philosophical Transactions of the Royal Society of London A: Mathematical, Physical and Engineering Sciences*, **365**(1859), pp. 2553–2585.
- [8] Athanasiadis, A., and Deconinck, H., 2003. “Object-oriented three-dimensional hybrid grid generation”. *International journal for numerical methods in engineering*, **58**(2), pp. 301–318.

-
- [9] Tomac, M., and Eller, D., 2014. "Towards automated hybrid-prismatic mesh generation". *Procedia Engineering*, **82**, pp. 377–389.
- [10] Dhopade, P., Capone, L., McGilvray, M., Gillespie, D., and Ireland, P., 2013. "Numerical modelling techniques for turbine blade internal cooling passages". In ASME Turbo Expo GT2015-42393.
- [11] Han, J.-C., 2013. "Fundamental gas turbine heat transfer". *Journal of thermal science and engineering applications*, **5**(2), p. 021007.
- [12] Smith, L., Karim, H., Etemad, S., and Pfefferle, W. C., 2006. "The gas turbine handbook".
- [13] Montenay, A., Paté, L., and Duboué, J., 2000. "Conjugate heat transfer analysis of an engine internal cavity". In Proceedings of Turbo Expo, no. 2000-GT-282.
- [14] Felippa, C. A., Park, K., and Farhat, C., 2001. "Partitioned analysis of coupled mechanical systems". *Computer methods in applied mechanics and engineering*, **190**(24), pp. 3247–3270.
- [15] Liu, Q., Luke, E. A., and Cinnella, P., 2005. "Coupling heat transfer and fluid flow solvers for multidisciplinary simulations". *Journal of Thermophysics and Heat Transfer*, **19**(4), pp. 417–427.
- [16] Verstraete, T., Alsalihi, Z., and Van den Braembussche, R., 2007. "Numerical study of the heat transfer in micro gas turbines". *Journal of turbomachinery*, **129**(4), pp. 835–841.
- [17] Sun, Z., Chew, J. W., Hills, N. J., Volkov, K. N., and Barnes, C. J., 2010. "Efficient finite element analysis/computational fluid dynamics thermal coupling for engineering applications". *Journal of turbomachinery*, **132**(3), p. 031016.
- [18] Coletti, F., Maurer, T., Arts, T., and Di Sante, A., 2012. "Flow field investigation in rotating rib-roughened channel by means of particle image velocimetry". *Experiments in fluids*, **52**(4), pp. 1043–1061.
- [19] Choi, Y.-H., and Merkle, C. L., 1993. "The application of preconditioning in viscous flows". *Journal of Computational Physics*, **105**(2), pp. 207–223.

- [20] Keshtiban, I., Belblidia, F., and Webster, M., 2004. “Compressible flow solvers for low mach number flows-a review”. *Int. J. Numer. Methods Fluids*, **23**, pp. 77–103.
- [21] Galperin, B., and Kantha, L., 1989. “Turbulence model for rotating flows”. *AIAA journal*, **27**(6), pp. 750–757.
- [22] Thompson, J. F., Soni, B. K., and Weatherill, N. P., 1998. *Handbook of grid generation*. CRC press.
- [23] Dhopade, P., Capone, L., McGilvray, M., Gillespie, D., and Ireland, P., 2015. “Numerical modelling techniques for turbine blade internal cooling passages”. In ASME Turbo Expo 2015, American Society of Mechanical Engineers, pp. GT2015–42393.
- [24] Ito, Y., 2013. “Challenges in unstructured mesh generation for practical and efficient computational fluid dynamics simulations”. *Computers & Fluids*, **85**, pp. 47–52.
- [25] Aftosmis, M. J., Berger, M. J., and Melton, J. E., 1998. “Robust and efficient cartesian mesh generation for component-based geometry”. *AIAA journal*, **36**(6), pp. 952–960.
- [26] Nakahashi, K., 2003. “Building-cube method for flow problems with broadband characteristic length”. In *Computational Fluid Dynamics 2002*. Springer, pp. 77–81.
- [27] Peskin, C. S., 2002. “The immersed boundary method”. *Acta numerica*, **11**, pp. 479–517.
- [28] Lohner, R., 1988. “Three-dimensional grid generation by the advancing front method”. *Int. J. Numer. Meths. Fluids.*, **8**, pp. 1135–1149.
- [29] Baker, T. J., 1989. “Automatic mesh generation for complex three-dimensional regions using a constrained delaunay triangulation”. *Engineering with Computers*, **5**(3-4), pp. 161–175.
- [30] Thompson, J. F., Soni, B. K., and Weatherill, N. P., 1998. *Handbook of grid generation*. CRC press.
- [31] Pirzadeh, S. Z., 2010. “Advanced unstructured grid generation for complex aerodynamic applications”. *AIAA journal*, **48**(5), pp. 904–915.
- [32] Lohner, R., 1993. “Matching semi-structured and unstructured grids for navier-stokes calculations”. *AIAA paper*, **93**, p. 3348.

-
- [33] Pirzadeh, S., 1994. “Unstructured viscous grid generation by the advancing-layers method”. *AIAA journal*, **32**(8), pp. 1735–1737.
- [34] Hassan, O., Morgan, K., Probert, E., and Peraire, J., 1996. “Unstructured tetrahedral mesh generation for three-dimensional viscous flows”. *International Journal for Numerical Methods in Engineering*, **39**(4), pp. 549–567.
- [35] Kallinderis, Y., and Ward, S., 1993. “Prismatic grid generation for three-dimensional complex geometries”. *AIAA journal*, **31**(10), pp. 1850–1856.
- [36] Kallinderis, Y., Khawaja, A., and McMorris, H., 1996. “Hybrid prismatic/tetrahedral grid generation for viscous flows around complex geometries”. *AIAA journal*, **34**(2), pp. 291–298.
- [37] Frey, P. J., and George, P.-L., 1999. *Maillages: applications aux éléments finis*. HERMES Science Publications.
- [38] Garimella, R. V., and Shephard, M. S., 2000. “Boundary layer mesh generation for viscous flow simulations”. *International Journal for Numerical Methods in Engineering*, **49**(1), pp. 193–218.
- [39] Bottasso, C. L., and Detomi, D., 2002. “A procedure for tetrahedral boundary layer mesh generation”. *Engineering with Computers*, **18**(1), pp. 66–79.
- [40] Alauzet, F., and Marcum, D., 2014. “A closed advancing-layer method with changing topology mesh movement for viscous mesh generation”. In *Proceedings of the 22nd International Meshing Roundtable*. Springer, pp. 241–261.
- [41] Ito, Y., and Nakahashi, K., 2002. “Unstructured mesh generation for viscous flow computations.”. In *IMR*, pp. 367–377.
- [42] Ito, Y., Shih, A. M., Soni, B. K., and Nakahashi, K., 2006. “An approach to generate high quality unstructured hybrid meshes”. *AIAA Paper*, **530**, p. 2006.
- [43] Connell, S. D., and Braaten, M. E., 1995. “Semistructured mesh generation for three-dimensional navier-stokes calculations”. *AIAA journal*, **33**(6), pp. 1017–1024.

- [44] Papadopoulos, I. V., 1998. “Critical plane approaches in high-cycle fatigue: on the definition of the amplitude and mean value of the shear stress acting on the critical plane”. *Fatigue & Fracture of Engineering Materials & Structures*, **21**(3), pp. 269–285.
- [45] Ito, Y., Shih, A. M., Soni, B. K., and Nakahashi, K., 2007. “Multiple marching direction approach to generate high quality hybrid meshes”. *AIAA journal*, **45**(1), pp. 162–167.
- [46] Aubry, R., and Lohner, R., 2009. “Generation of viscous grids at ridges and corners”. *International journal for numerical methods in engineering*, **77**(9), pp. 1247–1289.
- [47] Löhner, R., and Cebal, J., 2000. “Generation of non-isotropic unstructured grids via directional enrichment”. *International Journal for Numerical Methods in Engineering*, **49**(1-2), pp. 219–232.
- [48] Loseille, A., and Löhner, R., 2013. “Robust boundary layer mesh generation”. In *Proceedings of the 21st International Meshing Roundtable*. Springer, pp. 493–511.
- [49] Ito, Y., Murayama, M., Yamamoto, K., Shih, A. M., and Soni, B. K., 2011. “Efficient hybrid surface and volume mesh generation for viscous flow simulations”. *AIAA paper*, **3539**, p. 2011.
- [50] Marcum, D. L., 1996. “Unstructured grid generation components for complete systems”. In *5th International Conference on Grid Generation in Computational Fluid Simulations*, Starkville, MS.
- [51] Spalart, P., and Venkatakrishnan, V., 2016. “On the role and challenges of cfd in the aerospace industry”. *The Aeronautical Journal*, **120**(1223), pp. 209–232.
- [52] Bonet, J., and Peraire, J., 1991. “An alternating digital tree (adt) algorithm for 3d geometric searching and intersection problems”. *International Journal for Numerical Methods in Engineering*, **31**(1), pp. 1–17.
- [53] Shimrat, M., 1962. “Algorithm 112: position of point relative to polygon”. *Communications of the ACM*, **5**(8), p. 434.
- [54] Sutherland, I. E., Sproull, R. F., and Schumacker, R. A., 1974. “A characterization of ten hidden-surface algorithms”. *ACM Computing Surveys (CSUR)*, **6**(1), pp. 1–55.

-
- [55] Hormann, K., and Agathos, A., 2001. “The point in polygon problem for arbitrary polygons”. *Computational Geometry*, **20**(3), pp. 131–144.
- [56] Möller, T., and Trumbore, B., 1997. “Fast, minimum storage ray-triangle intersection”. *Journal of Graphics Tools*, **2**(1), pp. 21–28.
- [57] Tomac, M., and Eller, D., 2014. “Towards automated hybrid-prismatic mesh generation”. *Procedia Engineering*, **82**, pp. 377–389.
- [58] Vassberg, J., Dehaan, M., Rivers, M., and Wahls, R., 2008. “Development of a common research model for applied cfd validation studies”. In 26th AIAA Applied Aerodynamics Conference, p. 6919.
- [59] Tyssel, L., 2007. “The tritet grid generation system”. In Proceedings of the 10th International Conference on Numerical Grid Generation in Computational Field Simulations, International Soc. of Grid Generation (ISGG).
- [60] Han, J.-C., 2013. “Fundamental gas turbine heat transfer”. *Journal of thermal science and engineering applications*, **5**(2), p. 021007.
- [61] He, L., and Fadl, M., 2016. “Multi-scale time integration for transient conjugate heat transfer”. *International Journal for Numerical Methods in Fluids*.
- [62] Perelman, T., 1961. “On conjugated problems of heat transfer”. *International Journal of Heat and Mass Transfer*, **3**(4), pp. 293–303.
- [63] Luikov, A., Aleksashenko, V., and Aleksashenko, A., 1971. “Analytical methods of solution of conjugated problems in convective heat transfer”. *International Journal of Heat and Mass Transfer*, **14**(8), pp. 1047–1056.
- [64] Luikov, A., 1974. “Conjugate convective heat transfer problems”. *International Journal of Heat and Mass Transfer*, **17**(2), pp. 257–265.
- [65] Dorfman, A., and Renner, Z., 2009. “Conjugate problems in convective heat transfer: Review”. *Mathematical Problems in Engineering*, **2009**.
- [66] Maffulli, R., and He, L., 2014. “Wall temperature effects on heat transfer coefficient for high-pressure turbines”. *Journal of Propulsion and Power*, **30**(4), pp. 1080–1090.

- [67] Maffulli, R., and He, L., 2017. “Impact of wall temperature on heat transfer coefficient and aerodynamics for three-dimensional turbine blade passage”. *Journal of Thermal Science and Engineering Applications*, **9**(4), p. 041002.
- [68] Bohn, D., Bonhoff, B., Schonenborn, H., and Wilhelmi, H., 1995. “Prediction of the film-cooling effectiveness of gas turbine blades using a numerical model for the coupled simulation of fluid flow and diabatic walls”. *AIAA paper*(95-7105).
- [69] Han, Z., Dennis, B. H., and Dulikravich, G. S., 2001. “Simultaneous prediction of external flow-field and temperature in internally cooled 3-d turbine blade material”. *International Journal of Turbo and Jet Engines*, **18**(1), pp. 47–58.
- [70] Rigby, D. L., and Lepicovsky, J., 2001. “Conjugate heat transfer analysis of internally cooled configurations”. In *Proceedings of Turbo Expo*, no. 2001-GT-0405.
- [71] Sun, Z., Chew, J. W., and Hills, N. J., 2009. “Use of cfd for thermal coupling in aeroengine internal air systems applications”. In *Fluid Machinery and Fluid Mechanics*. Springer, pp. 399–404.
- [72] Verstraete, T., and Van den Braembussche, R., 2009. “A novel method for the computation of conjugate heat transfer with coupled solvers”. In *Proceedings of International Symposium on Heat Transfer in Gas Turbine Systems*.
- [73] Errera, M.-P., and Chemin, S., 2013. “Optimal solutions of numerical interface conditions in fluid–structure thermal analysis”. *Journal of Computational Physics*, **245**, pp. 431–455.
- [74] He, L., and Oldfield, M., 2011. “Unsteady conjugate heat transfer modeling”. *Journal of Turbomachinery*, **133**(3), p. 031022.
- [75] Batchelor, G. K., 1951. “Note on a class of solutions of the navier-stokes equations representing steady rotationally-symmetric flow”. *The quarterly journal of mechanics and applied mathematics*, **4**(1), pp. 29–41.
- [76] Batchelor, G. K., 2000. *An introduction to fluid dynamics*. Cambridge university press.

- [77] Giles, M. B., 1997. “Stability analysis of numerical interface condition in fluid-structure thermal analysis”. *International journal for numerical methods in fluids*, **25**(4), pp. 421–436.
- [78] Heselerhaus, A., 1998. “A hybrid scheme and stability analysis for coupled fluid/solid turbine blade temperature calculations”. In Proceedings of Turbo Expo, no. 98-GT-88.
- [79] Godunov, S. K., and Ryabenkii, V. S., 1997. *Difference schemes: an introduction to the underlying theory*. Elsevier.
- [80] Errera, M.-P., and Duchaine, F., 2016. “Comparative study of coupling coefficients in dirichlet–robin procedure for fluid–structure aerothermal simulations”. *Journal of Computational Physics*, **312**, pp. 218–234.
- [81] Sun, Z., Chew, J. W., Hills, N. J., Lewis, L., and Mabilat, C., 2012. “Coupled aerothermomechanical simulation for a turbine disk through a full transient cycle”. *Journal of Turbomachinery*, **134**(1), p. 011014.
- [82] Zhang, S., Chen, F., and Liu, H., 2014. “Time-adaptive, loosely coupled strategy for conjugate heat transfer problems in hypersonic flows”. *Journal of Thermophysics and Heat Transfer*, **28**(4), pp. 635–646.
- [83] Errera, M.-P., and Baque, B., 2013. “A quasi-dynamic procedure for coupled thermal simulations”. *International Journal for Numerical Methods in Fluids*, **72**(11), pp. 1183–1206.
- [84] Baque, B., Errera, M.-P., Roos, A., and Feyel, F. “Simulation of transient conjugate heat transfer via a temporal multiscale approach”. *International Journal for Multiscale Computational Engineering*, **11**(4), p. (2013).
- [85] Gimenez, G., Errera, M., Baillis, D., Smith, Y., and Pardo, F., 2016. “A coupling numerical methodology for weakly transient conjugate heat transfer problems”. *International Journal of Heat and Mass Transfer*, **97**, pp. 975–989.
- [86] 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 Proceedings of ASME Turbo Expo, no. GT2013-95426.
- [87] Chaquet, J. M., Corral, R., Gisbert, F., and Pastor, G., 2015. “A loosely coupled fluid/solid heat transfer method for disc cavities including mixing planes and a combination of 2d and 3d cavities”. In Proceedings of ASME Turbo Expo, no. GT2015-42269.
- [88] Corral, R., Crespo, J., and Gisbert, F., 2004. “Parallel multigrid unstructured method for the solution of the navier-stokes equations”. *AIAA Paper*, **761**.
- [89] Wilcox, D. C., et al., 1998. *Turbulence modeling for CFD*, Vol. 2. DCW industries La Canada, CA.
- [90] Gisbert, F., Corral, R., and Pastor, G., 2011. “Implementation of an edge-based navier-stokes solver for unstructured grids in graphics processing units”. In ASME Turbo Expo 2011, American Society of Mechanical Engineers, pp. gt2011-46224.
- [91] Burgos, M. A., Contreras, J., and Corral, R., 2011. “Efficient edge-based rotor/stator interaction method”. *AIAA journal*, **49**(1), pp. 19–31.
- [92] 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.
- [93] Corral, R., Pastor, G., and Contreras, J., 2011. “Proceedings of asme turbo expo”. In Turbomachinery Thermal Analysis Using Coupled Two and Three Dimensional Model and Reduced Order Fluid Models, no. GT2011-45998.
- [94] Verdicchio, J. A., Chew, J. W., and Hills, N. J., 2001. “Coupled fluid/solid heat transfer computation for turbine discs”. In Proceedings of Turbo Expo 2001, no. 2001-GT-0205.
- [95] Luikov, A., 1974. “Conjugate convective heat transfer problems”. *International Journal of Heat and Mass Transfer*, **17**(2), pp. 257–265.
- [96] Lindstedt, M., and Karvinen, R., 2013. “Conjugate heat transfer in a plate—one surface at constant temperature and the other cooled by forced or natural convection”. *International Journal of Heat and Mass Transfer*, **66**, pp. 489–495.

-
- [97] Eastwood, D., Coren, D., Long, C., Atkins, N., Childs, P., Scanlon, T., and Guijarro-Valencia, A., 2012. “Experimental investigation of turbine stator well , rim seal re-ingestion and interstage seal flows using gas concentration techniques and displacement measurements”. *Journal of Engineering for Gas Turbines and Power*, **134**(8), p. 082501.
- [98] Chaquet, J. M., Corral, R., Pastor, G., Pueblas, J., and Coren, D., 2011. “Validation of a coupled fluid/solid heat transfer method”. In Proceedings of ASME Turbo Expo, no. GT2011-45951.
- [99] Dixon, J. A., Valencia, A. G., Coren, D., Eastwood, D., and Long, C., 2014. “Main annulus gas path interactions-turbine stator well heat transfer”. *Journal of Turbomachinery*, **136**(2), p. 021010.
- [100] Chandrasekhar, S., 1961. *Hydrodynamic and Hydrodynamic stability*. OUP.
- [101] Tritton, D. J., 2012. *Physical fluid dynamics*. Springer Science & Business Media.
- [102] Kristoffersen, R., and Andersson, H. I., 1993. “Direct simulations of low-reynolds-number turbulent flow in a rotating channel”. *Journal of fluid mechanics*, **256**, pp. 163–197.
- [103] Smirnov, E., and Yurkin, S., 1983. “Fluid flow in a rotating channel of square section”. *Fluid Dynamics*, **18**(6), pp. 850–855.
- [104] Kheshgi, H. S., and Scriven, L., 1985. “Viscous flow through a rotating square channel”. *The Physics of fluids*, **28**(10), pp. 2968–2979.
- [105] Greitzer, E. M., Tan, C. S., and Graf, M. B., 2007. *Internal flow: concepts and applications*, Vol. 3. Cambridge University Press.
- [106] Cushman-Roisin, B., and Beckers, J.-M., 2011. *Introduction to geophysical fluid dynamics: physical and numerical aspects*, Vol. 101. Academic Press.
- [107] Tritton, D., 1992. “Stabilization and destabilization of turbulent shear flow in a rotating fluid”. *Journal of Fluid Mechanics*, **241**, pp. 503–523.
- [108] Bradshaw, P., 1969. “The analogy between streamline curvature and buoyancy in turbulent shear flow”. *Journal of Fluid Mechanics*, **36**(01), pp. 177–191.

- [109] White, F. M., and Corfield, I., 2006. *Viscous fluid flow*, Vol. 3. McGraw-Hill New York.
- [110] Wagner, J. H., Johnson, B. V., and Hajek, T. J., 1991. “Heat transfer in rotating passages with smooth walls and radial outward flow”. *Journal of Turbomachinery*, **113**(1), pp. 42–51.
- [111] Wagner, J., Johnson, B., and Kopper, F., 1991. “Heat transfer in rotating serpentine passages with smooth walls”. *Journal of Turbomachinery*, **113**(3), pp. 321–330.
- [112] Wagner, J., Johnson, B., and Kopper, F., 1991. “Heat transfer in rotating serpentine passages with smooth walls”. *ASME J. Turbomach*, **113**(3), pp. 321–330.
- [113] Di Sante, A., Theunissen, R., and Van den Braembussche, R. A., 2008. “A new facility for time-resolved piv measurements in rotating channels”. *Experiments in Fluids*, **44**(2), pp. 179–188.
- [114] Coletti, F., Jacono, D. L., Cresci, I., and Arts, T., 2014. “Turbulent flow in rib-roughened channel under the effect of coriolis and rotational buoyancy forces”. *Physics of Fluids*, **26**(4), p. 045111.
- [115] Mayo, I., Gori, G. L., Lahalle, A., and Arts, T., 2016. “Aerothermal characterization of a rotating ribbed channel at engine representative conditions-part i: High-resolution particle image velocimetry measurements”. *Journal of Turbomachinery*, **138**(10), p. 101008.
- [116] Mayo, I., Lahalle, A., Gori, G. L., and Arts, T., 2016. “Aerothermal characterization of a rotating ribbed channel at engine representative conditions-part ii: Detailed liquid crystal thermography measurements”. *Journal of Turbomachinery*, **138**(10), p. 101009.
- [117] Kays, W. M., Crawford, M. E., and Weigand, B., 2012. *Convective heat and mass transfer*. Tata McGraw-Hill Education.
- [118] Han, J., 1988. “Heat transfer and friction characteristics in rectangular channels with rib tabulators”. *Journal of Heat Transfer*, **1**, pp. 321–328.
- [119] Park, J., Han, J., Huang, Y., Ou, S., and Boyle, R., 1992. “Heat transfer performance

- comparisons of five different rectangular channels with parallel angled ribs”. *International Journal of Heat and Mass Transfer*, **35**(11), pp. 2891–2903.
- [120] Wright, L. M., Fu, W.-L., and Han, J.-C., 2005. “Influence of entrance geometry on heat transfer in rotating rectangular cooling channels ($ar=4:1$) with angled ribs”. *Journal of Heat Transfer*, **127**(4), pp. 378–387.
- [121] Huh, M., Lei, J., and Han, J.-C., 2012. “Influence of channel orientation on heat transfer in a two-pass smooth and ribbed rectangular channel ($ar=2:1$) under large rotation numbers”. *Journal of Turbomachinery*, **134**(1), p. 011022.
- [122] Taslim, M., and Spring, S., 1994. “Effects of turbulator profile and spacing on heat transfer and friction in a channel”. *Journal of Thermophysics and Heat Transfer*, **8**(3), pp. 555–562.
- [123] Johnson, B., Wagner, J., Steuber, G., and Yeh, F., 1994. “Heat transfer in rotating serpentine passages with trips skewed to the flow”. *Journal of Turbomachinery*, **116**(1), pp. 113–123.
- [124] Fu, W.-L., Wright, L. M., and Han, J.-C., 2004. “Heat transfer in two-pass rotating rectangular channels ($ar=1:2$ and $ar=1:4$) with 45 angled rib turbulators”. In ASME Turbo Expo 2004, American Society of Mechanical Engineers, pp. GT2004–53261.
- [125] Griffith, T. S., Al-Hadhrami, L., and Han, J.-C., 2002. “Heat transfer in rotating rectangular cooling channels ($ar=4$) with angled ribs”. *Journal of Heat Transfer*, **124**(4), pp. 617–625.
- [126] Liu, Y.-H., Wright, L. M., Fu, W.-L., and Han, J.-C., 2007. “Rib spacing effect on heat transfer in rotating two-pass ribbed channel ($ar=1:2$)”. *Journal of thermophysics and heat transfer*, **21**(3), pp. 582–595.
- [127] Liou, T.-M., Chang, S., Chen, J., Yang, T., and Lan, Y.-A., 2009. “Influence of channel aspect ratio on heat transfer in rotating rectangular ducts with skewed ribs at high rotation numbers”. *International Journal of Heat and Mass Transfer*, **52**(23), pp. 5309–5322.
- [128] Huh, M., Lei, J., Liu, Y.-H., and Han, J.-C., 2011. “High rotation number effects on

- heat transfer in a rectangular ($ar = 2:1$) two-pass channel”. *Journal of Turbomachinery*, **133**(2), p. 021001.
- [129] Iacovides, H., Jackson, D., Kelemenis, G., Launder, B., and Yuan, Y.-M., 2001. “Flow and heat transfer in a rotating u-bend with 45 ribs”. *International Journal of Heat and Fluid Flow*, **22**(3), pp. 308–314.
- [130] Rathjen, L., Hennecke, D., Sivade, C., and Semmler, K., 2002. “Detailed experimental and numerical heat/mass transfer investigations in a rotating two-pass coolant channel with staggered 45 ribs”. In 9th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC-9).
- [131] Viswanathan, A. K., and Tafti, D. K., 2006. “Large eddy simulation of fully developed flow and heat transfer in a rotating duct with 45 degree ribs”. In ASME Turbo Expo, American Society of Mechanical Engineers, pp. GT2006–90229.
- [132] Han, J.-C., and Chen, H.-C., 2006. “Turbine blade internal cooling passages with rib turbulators”. *Journal of Propulsion and Power*, **22**(2), pp. 226–248.
- [133] Han, J.-C., and Huh, M., 2010. “Recent studies in turbine blade internal cooling”. *Heat Transfer Research*, **41**(8).
- [134] Ligrani, P., 2013. “Heat transfer augmentation technologies for internal cooling of turbine components of gas turbine engines”. *International Journal of Rotating Machinery*, **2013**.
- [135] Mayo, I., Arts, T., El-Habib, A., and Parres, B., 2015. “Two-dimensional heat transfer distribution of a rotating ribbed channel at different reynolds numbers”. *Journal of Turbomachinery*, **137**(3), p. 031002.
- [136] Tanda, G., 2011. “Effect of rib spacing on heat transfer and friction in a rectangular channel with 45 angled rib turbulators on one/two walls”. *International Journal of Heat and Mass Transfer*, **54**(5), pp. 1081–1090.
- [137] Wang, Z., Corral, R., and Chedevergne, F., 2016. “Experimental and numerical study of heat transfer performance for an engine representative two-pass rotating internal cool-

- ing channel”. In ASME Turbo Expo, American Society of Mechanical Engineers, pp. GT2016–57419.
- [138] Rau, G., Cakan, M., Moeller, D., and Arts, T., 1998. “The effect of periodic ribs on the local aerodynamic and heat transfer performance of a straight cooling channel”. *Journal of Turbomachinery*, **120**(2), pp. 368–375.
- [139] Han, J.-C., and Zhang, Y., 1992. “Effect of uneven wall temperature on local heat transfer in a rotating square channel with smooth walls and radial outward flow”. *Journal of heat transfer*, **114**(4), pp. 850–858.
- [140] Parsons, J. A., Je-Chin, H., and Yuming, Z., 1994. “Wall heating effect on local heat transfer in a rotating two-pass square channel with 90 rib turbulators”. *International Journal of Heat and Mass Transfer*, **37**(9), pp. 1411–1420.
- [141] Zhang, Y., Han, J., Parsons, J., and Lee, C., 1995. “Surface heating effect on local heat transfer in a rotating two-pass square channel with 60 deg angled rib turbulators”. *Journal of turbomachinery*, **117**(2), pp. 272–280.
- [142] Hsieh, S.-S., Wang, Y.-S., and Chiang, M.-H., 1997. “Local heat transfer and velocity measurements in a rotating ribbed two-pass square channel with uneven wall temperatures”. *Journal of heat transfer*, **119**(4), pp. 843–848.
- [143] Al-Qahtani, M., Jang, Y.-J., Chen, H.-C., and Han, J.-C., 2002. “Prediction of flow and heat transfer in rotating two-pass rectangular channels with 45-deg rib turbulators”. *Journal of Turbomachinery*, **124**(2), pp. 242–250.
- [144] Al-Qahtani, M., Chen, H.-C., and Han, J.-C., 2003. “A numerical study of flow and heat transfer in rotating rectangular channels ($ar=4$) with 45 deg rib turbulators by reynolds stress turbulence model”. *Journal of Heat Transfer*, **125**(1), pp. 19–26.
- [145] Schüler, M., Zehnder, F., Weigand, B., von Wolfersdorf, J., and Neumann, S. O., 2011. “The effect of turning vanes on pressure loss and heat transfer of a ribbed rectangular two-pass internal cooling channel”. *Journal of Turbomachinery*, **133**(2), p. 021017.
- [146] Shevchuk, I. V., Jenkins, S. C., Weigand, B., von Wolfersdorf, J., Neumann, S. O., and

- Schnieder, M., 2011. “Validation and analysis of numerical results for a varying aspect ratio two-pass internal cooling channel”. *Journal of Heat Transfer*, **133**(5), p. 051701.
- [147] Schüler, M., Dreher, H.-M., Neumann, S., Weigand, B., and Elfert, M., 2012. “Numerical predictions of the effect of rotation on fluid flow and heat transfer in an engine-similar two-pass internal cooling channel with smooth and ribbed walls”. *Journal of Turbomachinery*, **134**(2), p. 021021.
- [148] Bonhoff, B., Parneix, S., Leusch, J., Johnson, B., Schabacker, J., and Bölcs, A., 1999. “Experimental and numerical study of developed flow and heat transfer in coolant channels with 45 degree ribs”. *International Journal of Heat and Fluid Flow*, **20**(3), pp. 311–319.
- [149] Saha, A. K., and Acharya, S., 2005. “Flow and heat transfer in internally ribbed ducts with rotation: an assessment of les and rans”. *Journal of Turbomachinery*, **127**(2), pp. 306–320.
- [150] Tafti, D. K., He, L., and Nagendra, K., 2014. “Large eddy simulation for predicting turbulent heat transfer in gas turbines”. *Philosophical Transactions of the Royal Society of London A: Mathematical, Physical and Engineering Sciences*, **372**(2022), p. 20130322.
- [151] Sewall, E. A., Tafti, D. K., Graham, A. B., and Thole, K. A., 2006. “Experimental validation of large eddy simulations of flow and heat transfer in a stationary ribbed duct”. *International journal of heat and fluid flow*, **27**(2), pp. 243–258.
- [152] Fransen, R., Vial, L., and Gicquel, L. Y., 2013. “Large eddy simulation of rotating ribbed channel”. In ASME Turbo Expo 2013, American Society of Mechanical Engineers, pp. GT2013–95076.
- [153] Menter, F. R., 1994. “Two-equation eddy-viscosity turbulence models for engineering applications”. *AIAA journal*, **32**(8), pp. 1598–1605.
- [154] Jenkins, S. C., Zehnder, F., Shevchuk, I. V., von Wolfersdorf, J., Weigand, B., and Schnieder, M., 2013. “The effects of ribs and tip wall distance on heat transfer for a vary-

ing aspect ratio two-pass ribbed internal cooling channel”. *Journal of Turbomachinery*, **135**(2), p. 021001.

- [155] Abdel-Wahab, S., and Tafti, D. K., 2004. “Large eddy simulation of flow and heat transfer in a 90 deg ribbed duct with rotation: Effect of coriolis and centrifugal buoyancy forces”. *Journal of turbomachinery*, **126**(4), pp. 627–636.