

Conjugate Heat Transfer in High Pressure Turbines



Roberto Maffulli
Hertford College
University of Oxford

A thesis submitted in partial fulfilment of the requirements for the degree of
Doctor of Philosophy

Hilary Term 2016

*To my mom, to my family.
A mia madre, alla mia famiglia.*

Acknowledgements

These acknowledgements are for all the people that contributed to this work and to my personal growth, for everyone who has been part of my (now good old) Oxford times, for the people that have always been part of my life since I was born, and for the person that joined me on the way at some point and changed everything.

My first and most sincere thank you goes to Prof. Li He for having given me the possibility to work on this thesis under his supervision. His suggestions, support, challenges and the fruitful discussions we had, taught me more than I could have learned from books and papers. He has been always available and receptive to my inputs, giving me freedom and guidance at the same time. I always appreciated your comments, never trivial: always sharp and straight. Thanks a lot for these four years.

A big thank you goes to Dr. Junsok Yi for the incredible help with Hydra and Fortran coding. I would not have made it without your advices.

If my lab hours were not too depressing it has also been merit of Cosimo, Aman-deep, Amir and lately Francesco (not to mention Dominos delivery guys). Coffee (and after work pints) would not have had the same taste without you. Thank you.

I would like to thank all the people that made my life (the other one...the one I had between a deadline and another, the life I happened to have when I realized that being more than 37 hours in a row in the lab without sleeping was not "life") in Oxford better. A big thank you goes to all of you: some of you I met for a year only, some stayed longer, some less, but you all I shared a piece of my life with. Thanks to you all.

Thanks to to Mark (the football champ), Tim (the squash champ), Alessandra (the fire alarm champ), Nora (the blanket champ), and all other people that made my first year in the MCR awesome. The MCR has not been the same once you left.

Being in the MCR committees for three years in a row has been a rewarding experience. A huge thank you goes also to all people involved in various levels in making the MCR a better place, from the MCR presidents and committees to the lodge staff. It has been a pleasure to be part of the Hertford family.

Thanks also to all the people that committed to the cause of waking up early on Saturday mornings, come to the pitch and invariably lose every single game Hertford MCR football team played in the four years I played. In the end victory is secondary if you can blame the captain isn't it?

Another huge thank goes to the Rocksonian crowd people: Jacob, Carla, Lluís, Daniel, Lorena, Spela, Roberto, Rosaria, Kirren, Audrey, Matteo, Martina, Marina and... last but not least Rocko. Thank you for being such great people. I am really missing all of you. A lot.

I would like also to thank the greatest zoo in the world: Bobby, Giorgio, Bufy, Ornella, Yak, Franco, Paolo, Francesca, Orso, Brigitte, Carmelo, Roberto, Arnaldo, Jessica and Abebe Joan. Best travel mates ever.

A special thank you goes to Cosimo. You have been a great office mate, still are an incredibly talented dancer, and will keep being a great friend. I am glad having shared so much with you.

This work is dedicated to my parents and my family. Without your efforts, words, and encouragements, I would not be writing these words. If I can walk by myself now, I owe that to you all.

Finally, the biggest thank you is for you: thanks Mari. You gave me all the support I could imagine for the choices I have made, and have been incredibly patient during my writing-up delirium and absence, but most importantly: you told me when to run, and we started running together. In you I have found a place to stay, the perfect place to stay.

Conjugate Heat Transfer in High Pressure Turbines

Roberto Maffulli

Hertford College

Supervised by Prof. L. He

A thesis submitted in partial fulfilment of the requirements for the degree of

Doctor of Philosophy

Hilary Term, 2016

Abstract

In the present thesis the link between aerodynamics and heat transfer in high pressure turbines is investigated through steady numerical calculations. The investigations include the effect of wall temperature on the Heat Transfer coefficient (HTC), aiming to understand whether the conventional assumption of HTC being invariant with the thermal boundary condition does hold in a typical compressible flow, where the aerodynamic and thermal fields are strongly coupled. A novel non-linear three point method is proposed to scale wall heat transfer accounting for the dependence of HTC on wall temperature and local flow history.

The effect of wall boundary condition on external aerodynamics and heat transfer calls for the need of Conjugate Heat Transfer (CHT) methods as design tools. For this reason CHT capabilities have been developed and integrated in Rolls-Royce Hydra CFD solver. The implemented CHT solver is fully-coupled, allowing for simultaneous solution of the solid and fluid domains. The implemented CHT coupling has been shown to be numerically stable with a good convergence rate for all cases tested. The implemented code has been successfully validated against both experimental, analytical and numerical data.

Conjugate analysis of a double-wall trailing edge cooling design has been performed under matched external Biot conditions. Aim of the investigation has been to quantify the effect of CHT on the cooling discharge characteristics and external aerodynamics in a cooling configuration where coolant and external flow are separated by a lower thermal resistance than in a traditional internal cooling configuration. Detailed CHT results for this case are presented and discussed.

Contents

I	Background	1
1	Introduction	2
1.1	Background	2
1.1.1	Trends and challenges in gas turbine development	2
1.1.2	Convective heat transfer in high pressure turbines	5
1.1.3	Conjugate heat transfer	7
1.2	Aims of the present work	8
1.3	Thesis outline	9
2	Literature Review	11
2.1	Effect of wall temperature on the heat transfer coefficient	12
2.1.1	Definition of an heat transfer descriptor invariant of T_w	13
2.1.2	Use of empirical correlations to correct Nusselt number	15
2.1.3	Use of a reference temperature to evaluate flow properties	16
2.2	Conjugate Heat Transfer	17
2.2.1	Effects of CHT in turbomachinery flows	18
2.2.2	Implementation strategies	23
2.3	Issues with CFD modeling of heat flux	25
2.3.1	Turbulence modeling	26
2.3.2	Transition modeling	27
2.4	Issues of interest to the present work	29
II	Investigation of the interaction between heat transfer and aerodynamics	32
3	Methods and validations	33
3.1	Baseline two-point HTC calculation procedure	34
3.2	Numerical method	37
3.2.1	Solution method	37
3.2.2	Fluid model	37
3.2.3	Computational domains and boundary conditions	38
3.2.3.1	Flat plate studies	38
3.2.3.2	MT1 studies	38
3.3	Flat plate heat transfer validation	39

3.4	MT1 Validation	42
3.4.1	2D MT1 validation	43
3.4.2	3D MT1 validation	45
3.5	Further numerical sensitivity studies	45
3.5.1	2D MT1 mesh dependency	47
3.5.2	3D MT1 mesh dependency	48
3.5.3	MT1 turbulence model sensitivity	48
3.5.4	HTC ΔT sensitivity	50
3.6	Final remarks	52
4	Wall temperature effect on HTC and aerodynamics	53
4.1	Boundary conditions	54
4.2	2D case studies	55
4.2.1	HTC dependence with temperature ratio	55
4.2.1.1	Further examination with CHT solution	58
4.2.2	Fluid driving temperature variation with TR	62
4.2.3	Thermal response of boundary layer profile	63
4.2.4	Influence of T_w on trailing edge flow shocks	66
4.3	3D case studies	68
4.3.1	HTC dependence with temperature ratio	68
4.3.2	Influence of TR on secondary flows	70
4.3.3	Influence of TR on passage mass flow	73
4.4	Influence of wall temperature on transition	74
4.5	Final remarks	78
5	An alternative working method for HTC	80
5.1	Comparison with existing correlation	81
5.2	The three-point method for heat flux scaling	83
5.3	Application to 2D blade section	85
5.4	Application to full 3D heat flux	86
5.5	Application to transitional flows	89
5.6	Concluding notes on the use of both two and three-point methods . .	90
5.7	Final remarks	91
III	Conjugate Heat Transfer development	93
6	Baseline RANS solver	94
6.1	Governing equations	95
6.2	Space discretization	96
6.2.1	Convective fluxes	99
6.2.2	Dissipative fluxes	100
6.2.3	Boundary conditions	102
6.3	Time discretization	104
6.4	Multigrid	106

6.5	Overview of Hydra workflow	106
7	Extension of fluid solver to Conjugate Heat Transfer solver	108
7.1	Governing equation and its discretization	108
7.2	Solid domain time-marching	109
7.3	Fluid-solid coupling strategy	110
7.3.1	Coupling strategy for corner nodes	113
7.4	Stability analysis	114
7.4.1	Solid solver	115
7.4.2	Coupling strategy	115
7.5	Further implemented features	116
7.5.1	Tripped transition modelling	117
7.5.2	Heat flux and convective thermal boundary conditions	122
8	Validation of the implemented solver	125
8.1	Computational domains and boundary conditions	126
8.1.1	Fluid and CHT studies	127
8.1.1.1	Flat plate	127
8.1.1.2	C3X Nozzle Guide Vane	128
8.1.2	Solid conduction studies	131
8.1.2.1	Hollow cylinder	131
8.1.2.2	L-shaped beam	132
8.2	Preliminary Hydra fluid-only validation	132
8.3	Validation of implemented thermal boundary conditions	134
8.3.1	Heat flux boundary condition	134
8.3.2	Convective heat flux thermal boundary condition	136
8.4	Solid solver validation	138
8.4.1	Hollow cylinder	138
8.4.2	L-shaped beam	139
8.5	Conjugate Heat Transfer solver validation	139
8.5.1	Flat plate with constant temperature at the bottom wall . . .	142
8.5.1.1	Laminar case	142
8.5.1.2	Turbulent case	145
8.5.2	Flat plate with constant heat flux at the bottom wall	147
8.5.2.1	Laminar case	147
8.5.2.2	Turbulent case	147
8.5.3	Flat plate with convective heat flux	150
8.5.3.1	Laminar case	150
8.5.3.2	Turbulent case	151
8.5.4	C3X nozzle guide vane	153
8.5.4.1	Mesh dependency	153
8.5.4.2	Validation results	156
8.6	Final remarks	158

9	CHT analysis of a cooled nozzle guide vane	160
9.1	Description of the trailing edge cooling	160
9.2	Similarity conditions	164
9.2.1	External Biot number matching	165
9.2.2	Internal Biot number matching	166
9.2.3	External to internal HTC's ratio matching	170
9.3	Computational domain and boundary conditions	171
9.4	Mesh dependency study	175
9.5	Results	176
9.5.1	Coolant discharge characteristics	179
9.5.2	Boundary layer traversing	185
9.5.2.1	Film cooling traversing	186
9.5.2.2	Suction side boundary layer traversing	189
9.5.3	Wall shear stress	194
9.6	Discussion and final remarks	195
10	Conclusions	198
10.1	Effect of T_w on HTC and aerodynamics	198
10.2	Non-linear HTC scaling method	199
10.3	Implementation and development of a CHT simulation environment	200
10.4	CHT solver validation	201
10.5	CHT evaluation of a double-wall trailing edge cooling system	202
10.6	Future works	203
A	The CHT pre-processor	205
A.1	The CHT preprocessor	205
A.1.1	Generation of the CHT coupling connectivity	205
A.1.1.1	Identification of planar and corner nodes	206
A.1.2	Mapping of the transition region	208
	Bibliography	212

List of Figures

1.1	Cut view of a high bypass ratio turbofan engine from Rolls-Royce (2015). Republished with permission of John Wiley and Sons.	3
1.2	Real Brayton cycle represented in a temperature (T) entropy (s) chart. 1-2: compression, 2-3: combustion, 3-4: expansion.	4
1.3	Development of engine pressure ratio over the years. Republished from Boyce (2012) with permission of Elsevier.	4
1.4	Development of engine firing temperature over the years. Republished from Boyce (2012) with permission of Elsevier.	4
1.5	Firing temperature increase with blade material improvement. Republished from Boyce (2012) with permission of Elsevier.	5
1.6	Typical cooling arrangement for a modern aero-engine Nozzle Guide Vane from Rolls-Royce (2015). Republished with permission of John Wiley and Sons.	6
3.1	Schematics of the differences between linear and non-linear $\dot{q} = f(T_w)$ relation when the h value is of interest.	36
3.2	Domain used for flat plate calculation with type of boundaries used. .	38
3.3	Domains used for MT1 calculations with type of boundaries used. . .	39
3.4	Calculations on laminar and turbulent boundary layer for constant T_w boundary condition: Nu comparison between Fluent and the reference correlation.	41
3.5	Grid used for the 2D calculations. Highlighted in red is the forced laminar flow region.	43
3.6	Aerodynamic validation for the 3D MT1 domain	44
3.7	Heat transfer validation for the 2D MT1 domain	45
3.8	Grid used for the performed 3D calculations.	46
3.9	Aerodynamic validation for the 3D MT1 domain.	46
3.10	Heat transfer validation for the 3D MT1 domain	47
3.11	Mesh dependency study for the 2D case: HTC distributions for the 0.6 TR case with different BL nodes count.	48
3.12	Mesh dependency study. Boundary heat flux (W/m^2) for the two mesh densities.	49
3.13	Turbulence model sensitivity study for both 2D and 3D MT1.	50
3.14	HTC distributions for $(T_{w+} - T_{w-})$ of 3 K, 5 K and 10 K.	51

4.1	Dependency of external HTC distribution on TR for 2D midspan calculations.	56
4.2	Dependency of external HTC distribution on TR using a Pr_t of 0.4 for 2D midspan calculations.	57
4.3	Dependency of external HTC distribution on TR using $k-\omega$ SST turbulence model Menter (1994) for 2D midspan calculations.	57
4.4	Calculated $\dot{q} = f(T_w)$ compared with linear approximation for the two wall points considered.	58
4.5	Mesh used for the conjugate solution.	60
4.6	Detail of metal temperature field for the 0.6 TR case.	61
4.7	Dependency of external HTC distribution on TR: comparison between conjugate and non-conjugate cases.	62
4.8	Dependency of T_{aw} distribution on TR for the 2D midspan results. . .	63
4.9	Re_{st} distributions across boundary layer at 5 traversing sections for 0.95 and 0.6 TR cases.	64
4.10	Traversing of turbulent viscosity μ_t distributions across boundary layer at leading edge and 95% pressure side for 0.95 and 0.6 TR cases. . . .	66
4.11	Suction side shock position for the quasi-adiabatic (TR=0.99) and cooled (TR=0.7) case.	67
4.12	HTC dependency on TR, on the vane pressure side.	69
4.13	HTC dependency on TR, on the vane suction side.	70
4.14	HTC dependency on TR, on the shroud endwall.	71
4.15	HTC dependency on TR, on the hub endwall.	71
4.16	Streamlines at suction side-endwalls corners for quasi-adiabatic (TR=0.99) and cooled (TR=0.7) case.	72
4.17	Wall shear stress direction on suction side for quasi-adiabatic (TR=0.99) and cooled (TR=0.7) case.	72
4.18	Wall shear stress magnitude on the late part of the suction side for quasi-adiabatic (TR=0.99) and cooled (TR=0.7) case.	73
4.19	Density contours at TE cut plane for quasi-adiabatic (TR=0.99) and cooled (TR=0.7) case.	74
4.20	$\dot{q}/\dot{q}_{ref}$ (Eqn. (3.6)) for TR=0.8 and 0.6 using the $k - kl - \omega$ transition model.	75
4.21	$\dot{q}/\dot{q}_{ref}$ (Eqn. (3.6)) for TR=0.8 and 0.6 using the transitional SST model.	75
4.22	Boundary layer traversing of laminar kinetic energy before the transition point for TR=0.8 and 0.6 using the $k - kl - \omega$ transition model.	76
4.23	HTC plots for TR=0.6 for $k - kl - \omega$ transition model, obtained using the traditional two-point method of Eqn. (3.2).	78
5.1	CFD results for $TR = 0.6$ case compared with Nu obtained using correlation bt Fit et al. Fitt et al. (1986).	82
5.2	Heat flux comparisons between the direct CFD solutions and those constructed from the 3-points model.	86
5.3	Heat flux comparisons between the direct CFD solutions and those constructed from the linear 2-points model.	87

5.4	Wall heat transfer (W/m^2) for $TR = 0.6$, direct CFD (center), conventional 2-point method (left) and new 3-point method (right).	88
5.5	Prediction error comparison between the two-point and three-point. .	88
5.6	Comparison of the new 3-point method and the conventional 2-point method for a transitional case using the $k - kl - \omega$ transition model.	89
6.1	Control volumes Ω_i and Ω_j (dashed) built around i-th and j-th mesh nodes.	98
6.2	Typical sawtooth mode.	102
6.3	Virtual node v , boundary node b , internal node i	104
6.4	General workflow of Hydra solver.	107
7.1	Fluid-solid interface nodes.	113
7.2	An example of corner node situation.	114
7.3	Tripped transition modeling.	118
7.4	T3A test case: tripped transition modeling with linear μ_t ramping. .	119
7.5	Schematics of the linear and non-linear ramping.	121
7.6	Final form of the ramping function used.	121
7.7	T3A test case: tripped transition modeling with non-linear μ_t ramping.	122
7.8	MT1 test case: tripped transition modeling with non-linear μ_t ramping.	123
7.9	Specification of wall temperature gradient through near-wall nodes. .	123
8.1	Computational domains and boundary types for the uncoupled and CHT Hydra flat plate calculations.	127
8.2	Computational domain and boundary types for the CHT simulations on C3X NGV.	129
8.3	Computational domain and boundary conditions for the solid simulations on the hollow cylinder case.	131
8.4	Computational domain and boundary conditions for the solid simulations on the L-shaped beam case.	132
8.5	Uncoupled calculations on laminar and turbulent boundary layer for constant T_w boundary condition: Nu comparison between Hydra, Fluent and the reference correlation.	133
8.6	Uncoupled calculations on laminar and turbulent boundary layer for constant $\dot{q}_w$ boundary condition: Nu comparison between Hydra, Fluent and the reference correlation.	135
8.7	Uncoupled calculations on laminar and turbulent boundary layer for convective heat flux boundary condition: Nu comparison between Hydra and Fluent.	136
8.8	Uncoupled calculations turbulent boundary layer for convective heat flux boundary condition: T_w comparison between Hydra and Fluent. .	137
8.9	Grid used for the cylinder solid conduction problem.	138
8.10	Temperature distribution in the cylinder: Hydra solid solver, ANSYS Mechanical and analytic solution.	139
8.11	Grid used for the corner solid conduction problem.	140

8.12	Comparison of temperature contours in the corner region of the L-shaped beam: Hydra solid solver and ANSYS Mechanical.	140
8.13	Grid used for the flat plate CHT calculations.	141
8.14	Flat plate CHT calculation, constant T_w case, laminar flow, thermal field in both the fluid and solid domain: comparison between Hydra CHT and Fluent.	143
8.15	Flat plate CHT calculation, constant T_w case, laminar flow: flat plate T_w comparison between Hydra CHT and Fluent.	144
8.16	Continuity of wall heat flux at fluid and solid side of the CHT interface.	144
8.17	Continuity of wall temperature at fluid and solid side of the CHT interface.	145
8.18	Flat plate CHT calculation, constant T_w case, turbulent flow, thermal field in both the fluid and solid domain: comparison between Hydra CHT and Fluent.	146
8.19	Flat plate CHT calculation, constant T_w case, turbulent flow: flat plate T_w comparison between Hydra CHT and Fluent.	147
8.20	Flat plate CHT calculation, constant $\dot{q}_w$ case, laminar flow, thermal field in both the fluid and solid domain: comparison between Hydra CHT and Fluent.	148
8.21	Flat plate CHT calculations for the constant $\dot{q}_w$ case: laminar flow. Wall temperature.	148
8.22	Flat plate CHT calculation, constant $\dot{q}_w$ case, turbulent flow, thermal field in both the fluid and solid domain: comparison between Hydra CHT and Fluent.	149
8.23	Flat plate CHT calculations for the constant $\dot{q}_w$ case: turbulent flow. Wall temperature.	149
8.24	Flat plate CHT calculation, convective boundary condition case, laminar flow, thermal field in both the fluid and solid domain: comparison between Hydra CHT and Fluent.	150
8.25	Flat plate CHT calculations for the convective heat flux case: laminar flow. Wall temperature.	151
8.26	Flat plate CHT calculation, convective boundary condition case, turbulent flow, thermal field in both the fluid and solid domain: comparison between Hydra CHT and Fluent.	152
8.27	Flat plate CHT calculations for the convective heat flux case: turbulent flow. Wall temperature.	152
8.28	Mesh used for CHT calculation on C3X NGV.	154
8.29	Mesh dependency study for C3X NGV: pressure distribution and wall non-dimensional HTC.	155
8.30	Mesh dependency study for C3X NGV: solid temperature distribution.	155
8.31	Results of the aerodynamic validation on the C3X NGV.	156
8.32	Results of the thermal validation on the C3X NGV: wall non-dimensional HTC.	157
8.33	Results of the thermal validation on the C3X NGV: non-dimensional T_w	157

9.1	Example of <i>double-wall</i> cooling scheme from Kvasnak et al. (2003).	162
9.2	Scheme of the trailing edge cooling design with relevant dimensions and main coolant flow paths (red).	163
9.3	Legend for the boundaries of the computational domain.	172
9.4	Details of the coarse and fine grids used for mesh independency tests.	176
9.5	Mach number contours for the <i>HM-HBR-CHT</i> case on the coarse and fine grids.	177
9.6	Temperature contours for the <i>HM-HBR-CHT</i> case on the coarse and fine grids.	177
9.7	Wall temperature distribution for the <i>HM-HBR-CHT</i> case on the coarse and fine grids. Temperature on walls 2, 3 and 4 is shown in a second graph for clarity as the temperature on these walls is markedly lower than the rest of the blade.	178
9.8	Nomenclature used for the coolant ejection holes.	180
9.9	Comparison of CHT Vs Adiabatic percentage difference (Eqn (9.18)) in <i>DR</i> , <i>VR</i> , <i>BR</i> and <i>I</i> for all coolant discharge sections (<i>F1</i> , <i>F2</i> , <i>F3</i> , <i>S</i>). Lower outlet Mach number case (<i>LM</i>).	182
9.10	Comparison of CHT Vs Adiabatic percentage difference (Eqn (9.18)) in <i>DR</i> , <i>VR</i> , <i>BR</i> and <i>I</i> for all coolant discharge sections (<i>F1</i> , <i>F2</i> , <i>F3</i> , <i>S</i>). Higher outlet Mach number case (<i>HM</i>).	182
9.11	Streamlines at the second film cooling ejection hole <i>F2</i> for the <i>LBR</i> and <i>HBR</i> cases. In red for the <i>LM-LBR</i> case is the limiting streamline between external hot gas and coolant flow.	184
9.12	Adiabatic film effectiveness for the first film ejection, <i>LM-HBR</i> case.	185
9.13	Positions and extension of film traversing cuts.	186
9.14	Distributions of <i>U</i> through the four traversing cuts <i>T1</i> , <i>T2</i> , <i>T3</i> , <i>T4</i> for the <i>LM - LBR</i> and <i>HM - LBR</i> cases.	187
9.15	Distributions of <i>U</i> through the four traversing cuts <i>T1</i> , <i>T2</i> , <i>T3</i> , <i>T4</i> for the <i>LM - HBR</i> and <i>HM - HBR</i> cases.	188
9.16	Distributions of ρU through the four traversing cuts <i>T1</i> , <i>T2</i> , <i>T3</i> , <i>T4</i> for the <i>LM - LBR</i> and <i>HM - LBR</i> cases.	190
9.17	Distributions of ρU through the four traversing cuts <i>T1</i> , <i>T2</i> , <i>T3</i> , <i>T4</i> for the <i>LM - HBR</i> and <i>HM - HBR</i> cases.	191
9.18	Distributions of <i>U</i> through the suction side traversing cut <i>SS</i> for the <i>LM - LBR</i> , <i>LM - HBR</i> , <i>HM - LBR</i> and <i>HM - HBR</i> cases.	192
9.19	Details of the position of <i>SS</i> cut with respect to the TE shock and the differences between adiabatic and CHT trailing edge Mach number contours. The results shown refer to <i>HM-HBR</i> case.	193
9.20	Distributions of ρU through the suction side traversing cut <i>SS</i> for the <i>LM - LBR</i> , <i>LM - HBR</i> , <i>HM - LBR</i> and <i>HM - HBR</i> cases.	194
9.21	Local values of wall shear stress at the positions of boundary layer traversing <i>T1</i> , <i>T2</i> , <i>T3</i> , <i>T4</i> and <i>SS</i> (see Fig. 9.13) for the <i>LM - LBR</i> , <i>LM - HBR</i> , <i>HM - LBR</i> and <i>HM - HBR</i> cases.	195

A.1	Workflow of the preprocessor for the generation of CHT coupling connectivity.	206
A.2	Wall planar and corner nodes mapping based on number of hexas sharing the same node N . Red edges connect the relevant nodes used for wall temperature update in the CHT solution.	207
A.3	Mapping of the transition region.	209

List of Tables

3.1	<i>Summary of the CFD boundary conditions used for flat plate validation.</i>	38
3.2	<i>Summary of the CFD boundary conditions used for 2D MT1 validation.</i>	39
3.3	<i>Summary of the CFD boundary conditions used for validation tests.</i>	40
4.1	<i>Summary of the wall temperatures used for the analyzed temperature ratios.</i>	54
4.2	<i>Summary of the internal cooling boundary conditions. Coolant bulk temperature T_b values are given in K, HTC h values in $W/(m^2K)$. Subscripts refer to the cooling hole number from LE to TE.</i>	60
6.1	Constants used for the 5-step Runge-Kutta time-marching.	105
7.1	Boundary conditions for the T3A flat plate case.	119
8.1	Boundary conditions used for Hydra flat plate calculations. The thermal boundary conditions are applied on the flat plate wall for uncoupled simulations and to the plate's bottom wall for CHT cases.	128
8.2	Flow conditions for the C3X experimental case No. 4311 (run 148) as from Hylton et al. (1983).	129
8.3	Boundary conditions imposed at inlet and outlet for the C3X validation case.	130
8.4	Boundary conditions imposed at cooling holes for the C3X validation case. The cooling holes numbers refer to Fig. 8.2.	130
8.5	Thermal properties of ASTM 310 from Facchini et al. (2004).	131
9.1	<i>Summary of dimensions of the trailing edge cooling design. Symbols are explained in Fig. 9.2.</i>	162
9.2	Flow conditions for the <i>LM</i> and <i>HM</i> cases.	173
9.3	Boundary conditions imposed at inlet and outlet for the <i>LM</i> and <i>HM</i> cases.	173
9.4	Pressure ratios at inlet of trailing edge cooling inlets for the <i>LBR</i> and <i>HBR</i> cases.	174
9.5	Values of HTC imposed at the plenum wall for all analyzed CHT cases.	175
9.6	Values for the coolant parameters at discharge sections for the adiabatic simulations.	181

Part I

Background

Chapter 1

Introduction

In this first chapter the background for the present research effort will be introduced. In the first sections a general introduction to the state of the art of gas turbines development trends, and challenges will be discussed. This will lead to introduce the importance of convective heat transfer modeling in high pressure turbines and the motivation for the Conjugate Heat Transfer (CHT) approach.

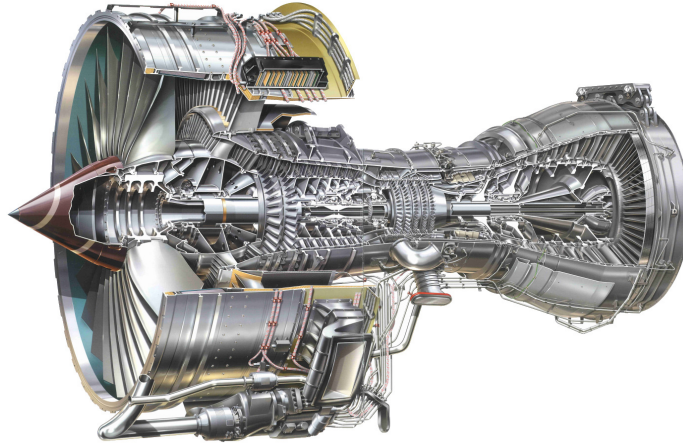
The motivations of the present work will then be summarized in Sec. 1.2. The chapter will be concluded with an overview for the present thesis in Sec. 1.3.

The contents of the first sections of this chapter do serve as a brief introduction to the issues that constitute the motivations of the present work but the discussion presented in this chapter is far from being fully exhaustive. A detailed review of the available literature for the topics that constitute the basis of this research work is presented in Ch. 2.

1.1 Background

1.1.1 Trends and challenges in gas turbine development

Gas turbines are one of the most commonly used systems for power generation both as propulsors in aircrafts as well as generators of mechanical work in land-based turbines for electrical power generation. Figure 1.1 shows a cut view of a typical high bypass



**Figure 1.1: Cut view of a high bypass ratio turbofan engine from Rolls-Royce (2015).
Republished with permission of John Wiley and Sons.**

ratio turbofan engine, commonly used in modern commercial airliners.

The conversion of thermal energy from the burning fuel to mechanical energy of the rotating shaft is realized in gas turbine engines through a Brayton cycle. The thermodynamic transformations constituting such cycle are shown in Fig. 1.2 in a T-s diagram. Air is compressed through a compressor (1-2), fuel combustion increases the temperature of the working fluid (2-3), the hot gases are then expanded in a turbine (3-4). The represented cycle considers irreversible transformations in both compression and expansion and is commonly referred to as *real Brayton cycle*, as opposed to the *ideal Brayton cycle*, considering isentropic expansion and compression. The efficiency and power output of a real Brayton cycle do depend on the compressor pressure ratio p_2/p_1 and on the temperature ratio T_3/T_1 , together with the turbine and compressor efficiencies (see Boyce (2012)). Particularly to an increase in T_3/T_1 (with constant p_2/p_1) corresponds an increase in both cycle efficiency and specific work output. Also, while increasing T_3/T_1 the pressure ratio required to maximize work output or efficiency tends to increase. The development trends of gas turbines reflect these considerations, with constantly increasing pressure ratios (see Fig. 1.3) and turbine entry temperatures T_3 (see Fig. 1.4. Despite the continuous improvement in turbine materials technology, the allowable blade metal temperature has increased

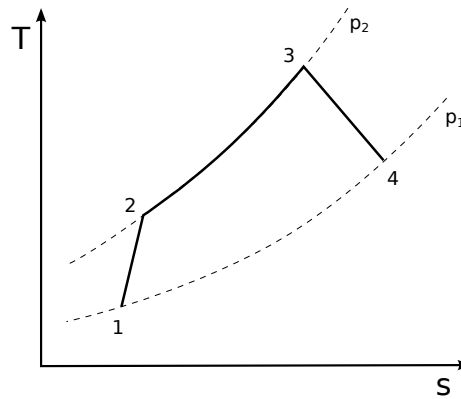


Figure 1.2: Real Brayton cycle represented in a temperature (T) entropy (s) chart. 1-2: compression, 2-3: combustion, 3-4: expansion.

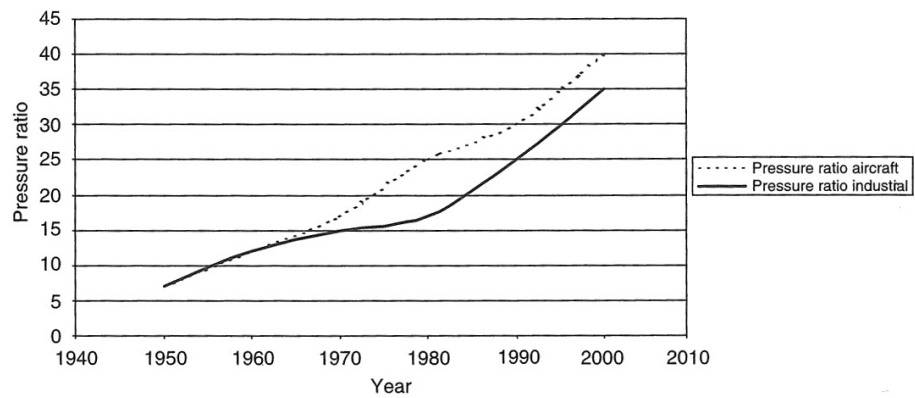


Figure 1.3: Development of engine pressure ratio over the years. Republished from Boyce (2012) with permission of Elsevier.

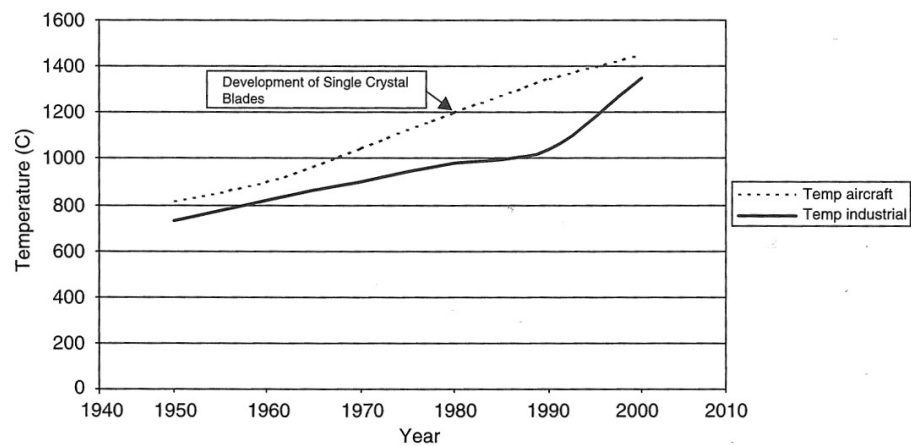


Figure 1.4: Development of engine firing temperature over the years. Republished from Boyce (2012) with permission of Elsevier.

by a mere 150 K in the past 50 years, while the turbine entry temperatures increased by 600 K in the same time (see Fig. 1.5). As a result of this unbalance between the material limits and the increase of hot gas temperatures current high pressure turbines run at temperatures far above the melting temperatures of the metal. The importance of turbine cooling has thus grown over the years and with it the necessity of correctly estimating the convective heat loads that turbine blades are subject to.

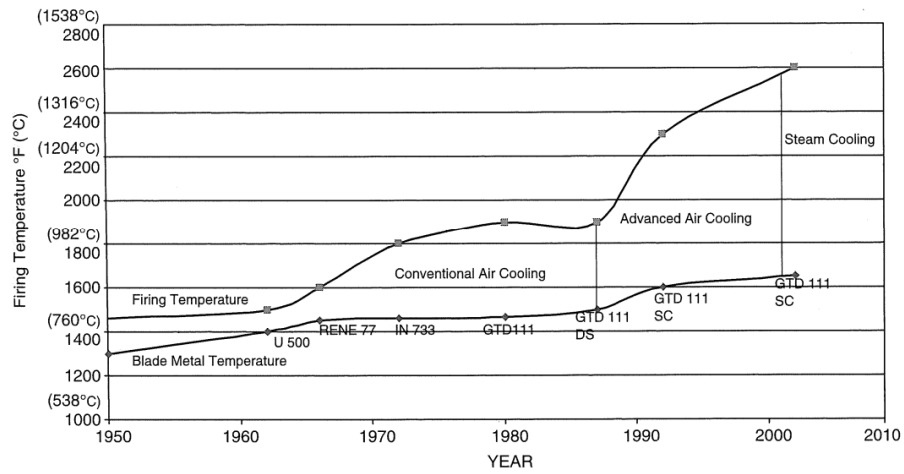


Figure 1.5: Firing temperature increase with blade material improvement. Republished from Boyce (2012) with permission of Elsevier.

1.1.2 Convective heat transfer in high pressure turbines

Figure 1.6 shows a typical cooling scheme for an aero-engine nozzle guide vane. The vane is cooled using air bled from the compressor. The design of internal cooling passages tries to maximize the amount of heat extracted by the cooling flow using a combination of cooling techniques (impingement, turbulators, pedestal banks). The cooling flow is then expelled through film cooling and showerhead holes, shielding the metal from the higher temperature external gas. Further protection of the metal from external gas is ensured by thermal barrier coating.

The blade metal temperature during service will depend on the energy balance between the internal cooling and external flow. A correct prediction of the convective

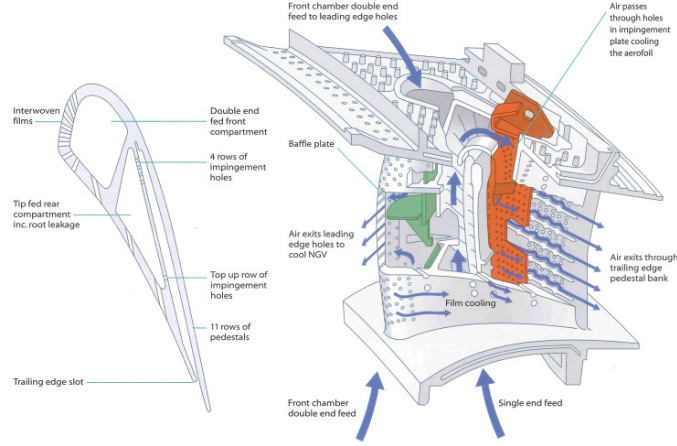


Figure 1.6: Typical cooling arrangement for a modern aero-engine Nozzle Guide Vane from Rolls-Royce (2015). Republished with permission of John Wiley and Sons.

phenomena involved is then of paramount importance for the correct design of the hot engine components. The issue of correctly modeling the convective heat flux in high pressure turbines becomes particularly important if one considers that 10 K difference in wall temperature can halve the life of a blade as reported by Han et al. (2012).

Convective heat transfer is described by Newton's law of cooling:

$$q = h (T_w - T_{ref}) \quad (1.1)$$

where h is the convective Heat Transfer Coefficient (HTC) and T_{ref} is an appropriate reference temperature. In the same way as Fourier's law of heat conduction expresses a linear relation between the heat flux and the driving temperature gradient, Newton's law of cooling introduces a linear relation between the driving temperature difference and the wall heat flux.

If the temperature field in the solid is of interest, one can imagine to predict it by applying a convective boundary condition (i.e. specifying h and T_{ref}) on the solid-fluid boundary and solving for the temperature field in the solid. This would result in the correct solid temperature if, and only if, the convective heat transfer descriptors

h and T_{ref} are not influenced by the wall temperature itself.

However, correct modeling of convective heat transfer is complicated by the fact that, differently from the thermal conductivity in Fourier's law, that is a physical property of the material, the convective heat transfer coefficient h is a flow property: it is closely connected with the aero-thermal field and its value does depend on the local flow conditions. Furthermore it is not always the case that the relation between convective heat flux and driving temperature is linear, as pointed out by Eckert & Drake (1971).

In the presence of compressible flows with high temperature gradients, typical of turbomachinery flows, the aerodynamic and thermal fields are closely coupled. Under those conditions the velocity field is influenced by the imposed heat transfer condition as highlighted also by Eckert & Drake (1971) and Kays & Crawford (1980). We should then expect the complications arising from modeling convective heat flux through h , that is flow-dependent, and on which the thermal field itself depends, in such conditions where it can be expected that a feedback of the thermal field on the aerodynamics may exist.

One can then expect that the thermal boundary condition would impact the flow field and, subsequently the convective heat transfer descriptors h and T_{ref} . In such a case the correct calculation of the solid temperature should pass by coupling the solid and fluid domains to account for the mutual interactions between the two: this approach is known as Conjugate Heat Transfer (CHT) and finds his full justification from the variability of the convective heat transfer descriptors with the thermal field itself. This is the fundamental aim of the first part of this thesis.

1.1.3 Conjugate heat transfer

A possible strategy to achieve a coupling between the two domains is to manually iterate from a CFD solver to a FEM structural solver, exchanging thermal boundary

conditions between the two. When solved in such an loosely coupled fashion, the solid and fluid regions may need to be solved several times until satisfying convergency is reached. Using Conjugate Heat Transfer the fluid and solid domains are considered in a simultaneously coupled fashion, rather than decoupled, enforcing continuity of heat flux and temperature at the interface between the two boundaries.

Using this approach the temperature distribution in the solid and at the fluid-solid interface is obtained as part of a single numerical solution. This allows to account for all the effects of the thermal field on the aerodynamics and on convective heat transfer, reflecting the physical interaction between the two domains.

The main advantage of using CHT is that it allows to obtain the thermal field in the solid domain, that is the final aim of the design process, allowing to bypass the uncertainties deriving from describing the convective heat transfer using HTC and allowing to eliminate costly manual iterations between a CFD code and a FEM code.

1.2 Aims of the present work

Given the discussed background the present work tries to address, through numerical studies, the following questions:

- In a typical compressible flow in a HP turbine, characterized by a strong coupling between aero-thermal fields, is the dependence of HTC on T_w negligible?
- Does the wall temperature have an impact on the aerodynamics?
- How does the thermal boundary condition and local flow history influence the HTC on the blade surface?
- How can one account for the dependency of HTC with wall temperature?

These questions are addressed in the first part of this thesis. The results do highlight the existing link between aerodynamics and heat transfer and underline the

need for CHT as a design tool. The second part of the thesis is focused on:

- Description of the implemented CHT capabilities in Rolls Royce Hydra CFD solver,
- Validation of the implemented solver,
- Using the developed CHT code to analyze a cooled NGV, with particular attention to the effects of CHT compared to adiabatic conditions.

1.3 Thesis outline

In chapter 2 the relevant literature is discussed. Particular attention is given to the current understanding regarding the variability of HTC with wall temperature and the effects of heat transfer on aerodynamics. The state of the art on CHT, its applications and the numerical implementations are discussed.

Chapter 3 presents the computational domains, numerical methods and sensitivity studies for the simulations performed in the first part of the thesis.

In chapter 4 the effect of wall temperature on HTC and aerodynamics on a 2D and 3D NGV configurations are discussed. The effects of wall temperature on secondary flows, shocks, passage mass flow are analyzed. A further discussion is given concerning the effect of wall temperature on transition and its implications on the definition and working of HTC.

Chapter 5 presents an alternative working method to scale the wall heat flux accounting for the dependency of HTC with wall temperature and local flow history.

The second part of the thesis begins with chapter 6, describing the numerical schemes implemented in the original Hydra flow solver.

In chapter 7 the implemented solver is described, including the fluid-solid coupling strategy. The stability of the coupling is discussed together with the additional

boundary conditions implemented in the code and an implemented tripped transition model.

Chapter 8 contains the results of the validation tests performed on the implemented CHT solver, together with the test cases performed to validate the solution of the solid conduction.

In chapter 9 the application of the developed solver to a cooled NGV is discussed.

The last chapter offers a summary of the work carried out and the corresponding conclusions.

In the appendix A the implemented CHT grid preprocessor is illustrated.

Chapter 2

Literature Review

A review of the available open literature relevant to the topics that will be covered in the following chapters is presented. This review serves to introduce the motivations of the present research effort, presented in Sec. 2.4.

A discussion concerning the effects of wall thermal boundary condition on HTC opens this chapter. In Sec. 2.2 a review of the relevant CHT studies in turbomachinery are presented followed by a survey of the current numerical options related to CHT numerical implementation.

A brief introduction to the limitations and active research areas in steady RANS modelling for turbomachinery will conclude the review.

In favour of clarity and organization of the contents, the part of the review concerning the effects of wall temperature on HTC and the CHT applications to turbomachinery has been split into two different sections. The author would like to emphasize, however, that the topics covered are deeply interconnected and, as such, this division is purely conceptual.

2.1 Effect of wall temperature on the heat transfer coefficient

A key quantity for the analysis of convective heat transfer is represented by the ratio between the conductive and the convective thermal resistances. The Nusselt number Nu quantifies this ratio and is defined as:

$$Nu = \frac{h(x)L}{k} = \frac{L}{T_f - T_w} \left(\frac{\partial T}{\partial y} \right)_w \quad (2.1)$$

From Eqn. (2.1) it is easy to see how Nu is equivalent to the non-dimensional value of the temperature gradient at the wall (Lakshminarayana (1995)):

$$Nu = \left(\frac{\partial(T/\Delta T_0)}{\partial(y/L)} \right)_w \quad (2.2)$$

Dimensional analysis of the energy equation (see Lakshminarayana (1995), Schlichting (1978)) brings to the following relation:

$$Nu = f(Re, Pr, Ec)^1 \quad (2.3)$$

It can be shown (Lakshminarayana (1995), Schlichting (1978)) that the Eckert number $Ec = U_\infty^2/(C_p \Delta T)$ can be expressed as:

$$Ec = (\gamma - 1)M^2 \frac{T_\infty}{\Delta T} = (\gamma - 1)M^2 \frac{1}{TR - 1} \quad (2.4)$$

Equations (2.3) and (2.4) show how the dependence of the Nusselt number on the wall TR can be inferred from simple dimensional analysis arguments.

In conditions where the dependence of HTC on the wall thermal boundary con-

¹The dependence on the Grashof number Gr in Eqn. (2.3) has been intentionally dropped as it is relevant only in natural convection flows.

dition is relevant, it introduces two inter-related problems for a reliable use of h as a heat transfer descriptor:

1. the heat transfer does not vary linearly with the driving temperature difference as expected,
2. using h as a heat transfer descriptor in the design phases can lead to non-optimal designs as the final heat load is dependent on the final wall temperature.

Different strategies have been developed to account or compensate for this dependence. The relevant literature for each of those will be discussed in the following sections.

2.1.1 Definition of an heat transfer descriptor invariant of T_w

Part of the scientific effort on the dependence of h on the wall thermal boundary condition has been dedicated to the quest for a definition that would make it insensitive to T_w , or to a complete bypass of the concept of HTC resorting to a different heat transfer descriptor².

Eckert & Drake (1971) proposed to define h based on an enthalpy difference, rather than a temperature difference showing analytically that it would be invariant with wall temperature. This argument however holds only under the assumption of laminar flow on a flat plate with $\rho\mu = \text{const.}$ and constant C_p , which is verified only where temperature differences between wall and mainstream are limited.

Moffat (1998) shows how the definition of HTC can be made independent from the thermal boundary condition using the adiabatic temperature $T_{adiabatic}$ as driving temperature. This, however, holds strictly only if no effect of the wall temperature on aerodynamics is observed. Furthermore $T_{adiabatic}$ is now dependent on the thermal

²Here the author refers to the use of Discrete Green's Functions (DGS) (see Moffat (1998)). The concept of DGS is mentioned for completeness but will not be deepened in the context of this literature review as it is of limited use in the turbomachinery community and shares the same limitations that will be clarified later regarding the $h_{adiabatic}$.

boundary condition and calculation of heat loads under a different wall temperature is quite cumbersome (see Moffat (1998), Anderson & Moffat (1992), Rhee & Moffat (2006)). Finally, the procedures established to calculate and use a HTC defined in such way rely on linear superposition theory, assuming that the energy equation is linear (Anderson & Moffat (1992), Moffat (1998)). The temperature levels often encountered in external turbine flows are such that non-linear effects are not negligible. As such $h_{adiabatic}$ is mostly used in internal flows.

To the author’s best knowledge, Gomes & Niehuis (2013) is the only relevant publication relative to the use of $h_{adiabatic}$ in external turbine flows and is only limited to a 2D case. The authors use the same methodology described in Moffat (1998) to obtain $h_{adiabatic}$ on a 2D cascade using CFD. From $h_{adiabatic}$ they then establish a procedure to obtain a conventionally defined HTC (Eqn. (1.1)) under arbitrary thermal boundary conditions. The paper clearly shows the complexity and limitations of such process and the authors final predictions of h show local differences up to 50% in HTC values when compared with experimental data. The same authors conclude that further investigation is needed to understand the sources of those discrepancies.

The previously discussed literature highlights how the definition of a heat transfer descriptor invariant with the wall thermal boundary condition is not universally established (though in some particular cases it can be found), and the use of $h_{adiabatic}$ has been of limited application so far in external turbine flows.

Typically for external high-speed heat transfer either the inlet temperature or the recovery temperature are used to define HTC: that makes h dependent on the thermal boundary condition. Typical approaches used to account for this dependency are the use of empirical correlations to correct Nu based on the wall to free-stream temperature ratio or the definition of a suitable reference temperature to evaluate flow properties. Both approaches are described in the next sections.

2.1.2 Use of empirical correlations to correct Nusselt number

A commonly used strategy to account for the variability of HTC with wall temperature is to consider a uniform scaling of the Nusselt number Nu of the form (Kays & Crawford (1980)):

$$Nu = Nu_0(TR)^n \quad (2.5)$$

where $TR = T_w/T_\infty$ is the ratio between wall and freestream temperatures and Nu_0 is the Nusselt number obtained with $TR = 1$.

Correlations of this form are typically used to scale experimental data available from low temperature conditions to engine-representative temperature ratios. The value of the exponent n does depend on the type of correlation used and the particular range of conditions the correlation has been obtained for.

Humble et al. (1951) proposes $n = 0$ for $TR < 1$ and $n = -0.55$ for $TR > 1$. Barnes (1960) presents the results of an experimental survey on the variability of HTC with wall temperature in internal flows for different gases, concluding that a value of $n = -0.4$ correlates best with the experimental data. Similarly Kays & Crawford (1980) suggest the use of $n = 0$ for $TR < 1$ and $n = -0.4$ for $TR > 1$. Eckert (1955) uses $n = -0.19$ for $0.6 < TR < 1.3$. Fitt et al. (1986) perform an analytical and experimental analysis on a compressible flat plate flow, correlating their data using $n = -0.25$ for $0.6 < TR < 1.3$. All the previously mentioned studies are confined to flat plate boundary layers or internal flows in pipes.

Hubbartt (1950) surveyed a set of cascade experimental data on a turbine reaction blade by Andrews & Bradley (1957). The blades were cooled to wall to free-stream temperature ratios ranging from 0.9 to 0.5. The authors tried to correlate the effect of TR on Nu and proposed the use of $n = -0.33$.

The use of correlations in the form of Eqn. (2.5) is the most commonly found in literature, but is not the only one. A conceptually similar approach is proposed by

Ziugzda & Zukauskas (1989). The authors, however, do not lump all variations of thermodynamic properties with wall temperature in a single exponent n but considers each parameter involved to be scaled to a single power law using:

$$Nu = Nu_0 \left(\frac{\mu_\infty}{\mu_w} \right)^n \left(\frac{k_\infty}{k_w} \right)^m \left(\frac{\rho_\infty}{\rho_w} \right)^p \left(\frac{C_{p_\infty}}{C_{p_w}} \right)^q \quad (2.6)$$

A more comprehensive review of available power-law correlations and their range of validity is presented by Brown & Donoughe (1950), Petukhov (1970) and, more recently, by Talib (2003). Correction methods in the form of Eqn. (2.5) are still object of research as the recent publication by Greiner et al. (2014) suggests. The authors proposed the use of a variable value of n , depending on the TR.

The scattering of the correction exponent n in the above mentioned correlations is emblematic of the empiricism involved in such correction methods. Moreover the amount of correction is strongly dependent on flow conditions. Kays & Nicoll (1963) and Fitt et al. (1986) note, for example, how laminar boundary layers seem to be less sensitive to TR compared to turbulent ones.

2.1.3 Use of a reference temperature to evaluate flow properties

Eckert (1955) introduced the possibility of evaluating the flow properties at a reference temperature T_f (Eqn. (2.7)) to obtain a value of Nu invariant of the wall thermal boundary condition. This approach has certainly proved to correlate effectively with a range of analytical or experimental data (Eckert (1955), VanFossen (1982), Kercher et al. (1982)) but does not seem to hold with a general validity. As an example Rued & Wittig (1986) concludes that Eckert method cannot account for the observed dependence of HTC on T_w for laminar flow on accelerating flat plate case, concluding that effect of wall temperature on flow properties is not the only cause of Nu variations

but effects on the aerodynamics should be accounted for.

$$\begin{cases} T_f = T_\infty + 0.5(T_w - T_\infty) + 0.22(T_r - T_\infty) \\ T_r = T_\infty \left(1 + \sqrt[3]{Pr} \frac{\gamma-1}{2} M_\infty^2 \right) \end{cases} \quad (2.7)$$

To account for the fact that different flow conditions may need a different definition of reference temperature Barnes (1960) proposed to use a reference temperature T_f defined as in Eqn. (2.8), where x is a factor to be tuned depending on flow conditions. This approach is analogous to what seen for the Nu correction correlations and just shifts the empirical effort from finding the value of the exponent n to finding x given a particular flow configuration.

$$T_f = T_\infty + x(T_w - T_\infty) \quad (2.8)$$

2.2 Conjugate Heat Transfer

The concept of Conjugate Heat Transfer is due to Perelman (1961), who first defined it, highlighted its motivations and provided the analytical solution of the conjugate problem for a set of simple incompressible flow cases.

Luikov et al. (1971) reported an analytical solution method for both steady and unsteady CHT in internal pipe flows and external flat plate. The proposed solution method considers both laminar and turbulent conditions as well as compressible flows. The same author introduces, in a following publication (Luikov (1974)) the concept of the Brun number³ as a conjugation parameter, concluding that where $Br > 0.1$ CHT effects should be accounted for. More recently Mosaad (1999) quantified the errors resulting from ignoring CHT effects on a laminar boundary layer flow. The

³The author defines the Brun number $Br \propto (k_f t)/(k_s x) Nu_x$ as a conjugation number. It can be easily seen how Br is proportional to the more commonly used Biot number Bi .

author concluded that errors in Nu exceeding 5% can be observed for $Br > 0.15$. It is expected that this error level would be higher in turbulent flows as Luikov (1974) points out.

Since its first definition and those first theoretical works, interest in CHT expanded continuously and its applications spread to different fields. Dorfman & Renner (2009) provided a thorough review of the available analytical CHT solutions to date and their relevant application fields.

In the following sections a review of the relevant literature regarding the applications of CHT to turbomachinery flows is presented. The review given here does not mean to be a fully exhaustive list of CHT applications in turbomachinery: particular attention will be given to the papers discussing the effects of CHT on aero/thermal flow conditions and typically held assumptions. The review will be followed by a survey of the typical numerical implementation strategies.

2.2.1 Effects of CHT in turbomachinery flows

One of the first conjugate studies related to turbomachinery external flows is the experimental and numerical campaign performed by Hylton et al. (1983). The authors analyzed the midspan heat transfer data of the C3X NGV and the Mark II rotor blade under varying external flow conditions and temperature ratios. The experimental database provided by the study is commonly used as a validation test for CHT codes. The results of the experimental campaign seem to suggest an increase of the local HTC with the decrease of the wall temperature but the observed effect falls in the experimental uncertainty for most of the blade surface. The reported data suggest also that a cooler wall seem to cause an earlier suction side transition on the C3X vane, though the observed effect may be a result of the experimental uncertainty.

Nirmalan & Hylton (1990) extended the work by Hylton et al. (1983), performing cascade experimental CHT analysis on the C3X vane equipped with film and

internal cooling. They investigated also the effects of the coolant to mainstream temperature ratio on film cooling HTC . The authors identified two aero/thermal key aspects responsible of the evolution of the film HTC on the blade surface: the T_c/T_g temperature ratio itself and its effect on the blowing ratio and on film turbulence. The authors showed that both aspects have opposite effects on local HTC levels and their relative magnitude depends on external flow conditions but, in a typical case, the effect of the coolant to mainstream temperature ratio is far from negligible. The appreciation of such effects is particularly relevant when compared with the common film cooling experimental practice of density ratio matching through the use of foreign gases (Jones (1999), Day et al. (2000)).

Na et al. (2007) performed a computational CHT analysis of the effect of TBC on the thermal gradients in the solid domain for a flat plate with internal cooling and film ejection. The authors showed that CHT effects should be accounted for in order to correctly understand the protection offered by the film. They showed that, without TBC, the metal conduction decreases the effect of hot gas entrainment after the cooling hole. The inclusion of TBC reduces the metal temperature but alters its thermal stresses introducing higher temperature gradients. The authors also observe changes in the 3D film velocity field when altering the thermal boundary condition. Non negligible changes in the film secondary flows between adiabatic and CHT solution have been reported also by Bohn et al. (2003).

Williams et al. (2014) performed an experimental evaluation of the overall effectiveness considering both internal impingement cooling and suction side film coverage. The experimental analysis was carried out on a Biot matching model. The authors showed that for high blowing ratios, despite the low effectiveness due to film detachment from the wall the overall effectiveness was still high due to the internal convection. The consequence of the coupling between internal cooling and external film ejection was that the optimal flow rate resulting in the lowest wall temperatures

(both peak and average) and temperature gradients was shown to be much higher than would be expected considering only the adiabatic effectiveness as a performance parameter. Also worth mentioning is the observation by the authors that a difference in overall effectiveness as little as 0.02 equates to a change in engine temperatures on the order of 30 K : enough to halve part life for temperature driven failure modes Han et al. (2012). In this view the importance of accounting for CHT effects in a realistic way becomes crucial.

Harrison & Bogard (2008*b*) performed a computational analysis of a film cooled flat plate. Mimicking typical experimental practice, the authors obtained the values of film HTC imposing a constant heat flux at the fluid-solid interface. Adiabatic wall temperature distribution was then obtained through an adiabatic simulation. The authors then used HTC and T_{aw} to calculate the corresponding wall heat flux. An uncoupled thermal simulation of the solid domain was also performed to obtain the temperature distribution. Both the so-obtained heat flux and temperature distributions were compared to the ones resulting from a fully coupled CHT analysis. The authors reported discrepancies in overall film effectiveness up to 10%. Furthermore they have shown that agreement between heat flux predictions from the uncoupled analysis and the CHT one correlated well with how closely the adiabatic wall temperature approximated the over-riding gas driving temperature for heat transfer into the wall.

Authors from the same research group (Dobrowolski et al. (2010)) extended the work carried out by Harrison & Bogard (2008*b*) to investigate the validity of conventionally defined heat transfer augmentation factors and adiabatic wall temperature on a film cooled turbine leading edge, finding differences in wall heat flux up to 30% between CHT and conventional uncoupled calculations. The authors do not report the corresponding differences in overall effectiveness.

Dyson et al. (2014) present an experimental and computational CHT analysis of

a film cooled vane. The authors concluded, similarly to previous studies that the spatial variations of the temperature field in a conjugate case are much lower than the film effectiveness contours would suggest. Furthermore the thermal interaction between the film flow and the walls is such that the overlying gas temperature is always cooler than the adiabatic wall temperature for detached film. This situation was reversed for attached films under lower blowing ratios due to the increased heat pickup of the film flow. The issue of the use of T_{aw} as driving temperature has been examined also by Dees et al. (2011) and later by Stewart & Bogard (2015).

In Dees et al. (2011), the authors seem also to note an earlier reattachment of a detached film on the adiabatic case with respect to CHT and attribute this to differences in momentum flux ratio between the two cases. The authors do not investigate the issue further.

Nathan et al. (2014) analyzed the results of an experimental campaign on an NGV leading edge with internal impingement cooling and showerhead. Their results highlight that, in the showerhead region, the adiabatic effectiveness seems to be a reliable indicator of the presence of hot spots in the conjugate model. Earlier results by Heidmann et al. (2000), however, suggest that this observation depends on the particular local aero/thermal conditions. In Heidmann et al. (2000) a computational analysis of a fully cooled NGV with showerhead and film ejection on pressure side and suction side is carried out. The analysis of the showerhead region heat transfer data reveals that there are regions where, despite the high values of adiabatic effectiveness, the turbulent interaction between film jets is such that the increase in local HTC reveals high levels of wall heat flux.

Attention to CHT is not limited to turbine external flows. Conjugate analysis of internal cooling configuration is also an active research field.

Fedrizzi & Arts (2004) and Agostini & Arts (2005) performed experiments on an internal channel with inclined and straight ribs respectively. Both analyses revealed

considerable differences in the inter-rib Nu distributions with Agostini & Arts (2005) reporting a 35% increase in the mean enhancement factor between a pure convective and a conjugate experiment.

Coletti et al. (2010) performed experimental analysis on a ribbed trailing edge cooling slot. The author reported differences in average Nu up to 25% between the uncoupled and CHT case, resulting in a mismatch of predicted wall temperature up to 30 K. Similar conclusions have been drawn by Cukurel et al. (2012) and Cukurel & Arts (2013). The authors in Cukurel & Arts (2013) document also an earlier reattachment (half of the rib height) of the coolant flow past the rib in the purely convective case with respect to CHT conditions. A key conclusion from those studies is that the general redistribution of the heat in the rib surface is influencing the local conditions in a way that is inherently dependent on flow history and, as such, hardly possible to account for without recurring to CHT analysis.

The bottom line aim of a turbine thermal analysis would be to evaluate a design and predict its life expectancy given the external heat load and cooling design. A few authors discussed the implications of the dependence of Nu on the thermal boundary condition with respect to life prediction.

Ramachandran & Shih (2015) discussed the importance of the Biot number (Bi) analogy in design of experiments showing that Bi is a dominant factor when meaningful wall temperature distributions are of interest. The importance of matching Bi becomes fundamental when thermal stresses in the solid and life expectancy predictions are of interest. Recently CHT has been used to perform thermal stress analysis and life expectancy predictions of turbine blades by Mazur et al. (2006), Amaral et al. (2010), Verstraete et al. (2010) and Dawes et al. (2010). The benefit of the use of more conductive materials to decrease thermal stresses in the metal has been discussed by Montomoli et al. (2012).

Starke et al. (2008), and more recently Caloni & Shahpar (2015) analyzed the

errors in the wall temperature prediction on a turbine blade, comparing a CHT analysis and a more classical uncoupled analysis. In the uncoupled analysis, the HTC and driving temperature data from aerodynamic computations are used to predict the temperature distribution in the metal through FEM analysis. In both cases the authors showed that the differences in temperature predictions are far from negligible. Interestingly Caloni & Shahpar (2015) adopted the novel non-linear method that that was proposed and developed in this thesis work (see Ch. 5), showing that its application dramatically improves the temperature prediction in the solid in an uncoupled approach.

2.2.2 Implementation strategies

Numerical implementation strategies are normally classified into two categories, depending on whether the simulation is carried out using a single solver or two: one for the fluid and the other for the solid domains. In the following discussion the methods belonging to the first category will be referred to as *fully coupled* while the term *loosely coupled* will be used for the second.

In loosely coupled methods two different solvers are used in the solid and fluid domains and a continuous exchange of boundary conditions at the solid-fluid interface is performed throughout the simulation until convergence. Conversely fully coupled methods use a single solver, removing the need for a constant exchange of interface boundary conditions.

Heselhaus (1998) discusses the advantages/disadvantages of both approaches. Loosely coupled methods allow for a greater flexibility and can exploit the strengths of two different solvers in the two different domains. As an example Heidmann et al. (2003) performed a CHT analysis of a film cooled turbine vane using a BEM solver to calculate the thermal field in the solid domain. This allowed to obtain a CHT solution without meshing the interior of the solid domain as BEM methods only require the

boundaries to be meshed. More commonly a coupling between a FV fluid solver and a FEM solver in the solid is realized (see Amaral et al. (2010), Illingworth et al. (2005) and Wlassow et al. (2010)).

The interfacing of two solvers requires, however, costly iterations. The different mesh requirements in the solid and fluid domains require to interpolate the nodal values from one domain to another (Verstraete (2008)) introducing an approximation that does not guarantee energy conservation at the interface. Finally, the presence of aero/thermal features showing high sensitivity to wall temperature variations (e.g. transition), can have a negative impact on the convergence history of loosely coupled methods.

Several strategies are commonly used in loosely coupled methods, depending on the type of boundary condition (temperature, heat flux, HTC and driving temperature) exchanged from one domain to the other and viceversa. Verstraete & Van den Braembussche (2009) describe the stability criteria for loosely coupled methods based on the type of boundary conditions passed at the interface, showing that none of the methods is unconditionally stable. Stability for loosely coupled methods depends on the Biot number of the conjugate case analyzed and the number of iterations performed in the fluid domain per each coupling step.

In fully coupled methods a single solver is used to solve both fluid and solid domains. Despite requiring modifications to the original fluid solver, this approach allows for an implicit conservation of energy at the fluid-solid interface. Furthermore it can be shown (Giles (1997)) that fully coupled methods are unconditionally stable. Examples of fully coupled CHT implementations can be found in Rigby & Lepicovsky (2001) and Montomoli et al. (2004).

A fundamental problem of CHT modeling is related to the time scales mismatch between the solid and fluid domains. He & Oldfield (2011) show that the thermal response in the solid domain can be estimated being four orders of magnitude slower

than the one in the fluid side for a typical turbomachinery application. This brings to a considerable problem when unsteady CHT simulations are of interest as a time-marching based on the solid time scale would be unpractical in most situations.

Conventional approach used to model unsteady CHT is to use a loosely coupled method assuming a steady state for the fluid domain (Illingworth et al. (2005), Sun et al. (2008)).

He & Oldfield (2011) proposed an unsteady CHT methodology in which the fluid domain is solved in the time domain, while the solid is in the frequency domain, allowing to solve the unsteady problem keeping the time consistency.

2.3 Issues with CFD modeling of heat flux

Steady RANS is nowadays a commonly used tool in turbomachinery design. Despite the considerable improvements, the accuracy of heat transfer predictions obtainable by computational analysis is still lagging with respect to the accuracy obtainable when aerodynamic data are of interest (Dunn (2001), Tallman et al. (2009)).

A key aspect related to a reliable heat flux modelling is the choice of the wall refinement. Heat transfer problems require a considerably finer grid at the wall due to the need to fully resolve the thermal boundary layer and the wall temperature gradient (Dunn (2001)). The numerical study by Huang et al. (1997) on flat plate boundary layer predictions showed that different turbulence models seem to have different sensitivities with respect to the loss in accuracy deriving from the use of wall functions, with the Spalart-Allmaras turbulence model being the most tolerant among the investigated ones. Chi & Shih (2012) showed that non-negligible differences in wall Nu can arise if wall-functions are used even for a simple case of laminar flow in a pipe. In Tolpadi et al. (2005) the authors present CFD results on a NGV, showing that the use of wall functions for heat transfer calculations can considerably alter the HTC

distribution (differences up to 50% with respect to experimental data were observed). Despite there seems to be a high consensus regarding the use of a low-Reynolds approach when heat transfer problems are of interest, recent research by Mazzoni et al. (2015) suggests however that reasonably good predictions in turbomachinery flows can be obtained also with the use of wall functions.

Recent reviews by Denton (2010) and Horlock & Denton (2005) discussed the limitations and strengths of CFD. Both authors highlighted how turbulence and transition models are key factors contributing to the uncertainties of CFD results. This is true for both aerodynamic as well as heat transfer predictions.

The main limitations of current state of the art of turbulence and transition modelling capabilities will be discussed in the following sections. The discussion will be restricted to the turbulence models based on the Boussinesq hypothesis.

2.3.1 Turbulence modeling

Typical turbulence models used in turbomachinery applications are: Spalart-Allmaras (Spalart & Allmaras (1992)), $k - \epsilon$ in its various forms (standard Launder & Spalding (1974), RNG Yakhot & Orszag (1986) and realizable Shih et al. (1994)) and $k - \omega$ (standard Wilcox et al. (1998) and SST Menter (1994)).

One of the main issues related to the use of two-equations turbulence models is the excessive production of turbulent kinetic energy k in presence of large strain rates. This is usually referred to as *the leading edge anomaly*. The overproduction of turbulent kinetic energy does not only impact the solution in regions of high strain rate: the effects of the leading edge anomaly can be seen in other parts of the computational domain due to the transport of k . Medic & Durbin (2002) showed, how commonly used two-equations turbulence models can result in markedly wrong heat transfer predictions on the whole blade surface due to an excessive production of turbulent kinetic energy at the leading edge. The author shows that the use of appropriate

limiters can damp the unrealistic production of k .

A second, typically known, limitation related to the use of classical turbulence models based on Bussinesq hypothesis is the poor accuracy shown by these models in predicting the lateral spreading of jets in crossflow (see Walters & Leylek (1997) and more recently Harrison & Bogard (2008*a*) and Dyson et al. (2014)). This is particularly relevant where film cooling coverage predictions are of interest. Walters & Leylek (1997) attributed the lack of diffusion of the film flow to the inherent assumption of isotropy of turbulence in classical turbulence models. Mangani & Andreini (2008) showed that modification of the turbulence model to include a non-isotropic value for μ_t , as proposed by Bergeles et al. (1978), can dramatically improve the predictions of film diffusion.

Another key aspect relative to the jet diffusion is the turbulent Prandtl number as its value directly affects the local values of thermal turbulent diffusivity. Traditional turbulence modelling approach considers Pr_t to be constant, but this assumption has been shown to be inadequate both in simple boundary layer flows (Kays (1994)) and in film cooling flows (Kohli & Bogard (2005)). The use of turbulence models that could include a spatial variation of Pr_t has shown to improve the predictive performances of the lateral spreading of jets (see Liu et al. (2008)).

2.3.2 Transition modeling

Bypass transition is the most common mechanism of transition in turbomachinery (Mayle (1991)). Bypass transition owes its name to the fact that it is typically present in flow conditions where external disturbances to the boundary layer (e.g. turbulence levels) are so high that transition is promoted bypassing the classical instability mechanisms present in natural transition.

The mechanism of bypass transition has been very clearly described by Walters & Cokljat (2008). When freestream turbulence intensities are low, typically less than

1%, the development of velocity fluctuations that are precursor of transition is dominated by self-sustained instability mechanisms: the Tollmien-Schlichting instabilities (Schlichting (1978)). In presence of considerable external turbulence, however, the mean velocity in the boundary layer becomes increasingly distorted, with a higher momentum in the inner region and a decrease of momentum in the outer Matsubara & Alfredsson (2001). To the change in velocity in the boundary layer in the pre-transitional region corresponds the appearance of streamwise fluctuations, due to the presence of low frequency streamwise vortices in the boundary layer. These bring ultimately to transition and are initiated by the work of external pressure disturbances in the boundary layer flow (Mayle & Schulz (1996)).

Accurate prediction of transition (both in terms of transition onset point and end) is fundamental both for aerodynamic performance (this is particularly true for low pressure turbine stages) and heat transfer predictions (this is fundamental for high pressure turbine blades). It is in fact well known that the wall Nu is a strong function of the boundary layer status (Kays & Crawford (1980)).

Despite the progressive improvements in transition modelling, still one of the most commonly used approaches in industrial and research practice (Denton (2010), He & Oldfield (2011)) is to fix the transition point a priori and artificially tripping the boundary layer transition in that point. This approach has the advantage of simplicity and does not need additional transport equations to be solved on top of the turbulence model used. The drawback of such approach is that it models transition through a non-physical sharp jump from a laminar to a turbulent state and that the transition point is not determined as part of the solution but imposed at the beginning in a way that relies completely on the user.

More sophisticated models are commonly used to model transition, the most popular of them are the $k - k_l - \omega$ (Walters & Cokljat (2008)) and the $\gamma - Re_\theta$, also called transitional SST model (Menter et al. (2006)).

The $k - k_l - \omega$ transition model is based on the observation that bypass transition is promoted by streamwise flow oscillations having a different behaviour than turbulent fluctuations. Mayle & Schulz (1996) first proposed to model these fluctuations through the definition of a laminar kinetic energy k_l . In the $k - k_l - \omega$ model a transport equation for k_l is solved together with equations for k and ω . Conceptually the model marks the onset of transition when the ratio between the turbulent production time-scale and the molecular diffusion time-scale exceeds a defined value. Transition is modelled through a transfer of laminar kinetic energy to turbulent kinetic energy. The advantage of this method is that the turbulence closure is based purely on physical considerations and does not need much empirical input.

The $\gamma - Re_\theta$ model is, differently from the $k - k_l - \omega$, a correlation based model. The model is based on the SST formulation of $k - \omega$ turbulence model, but solves two extra equations: one for the intermittency (γ) and the second for the Reynolds number based on the transition onset momentum thickness (Re_θ). The intermittency is used to trigger the production of turbulent kinetic energy in the boundary layer, while Re_θ is needed to mark the start of the transition. The model uses a correlation that provides the value of the transition Re_{θ_t} outside of the boundary layer, as function of the external flow conditions (Tu , pressure gradients). This value is then diffused in the boundary layer using a transport equation for Re_{θ_t} . In the intermittency equation, the ratio between local values of momentum thickness Re_θ and the transition Re_{θ_t} is used to mark the inception of transition.

2.4 Issues of interest to the present work

The discussion regarding the current understanding on the effects of wall temperature on HTC has showed that considerable scientific effort has been lavished in the effort of accounting for such dependence. It is however true that, despite the convenience

of adopting simple corrections of the kind of the ones shown in Sec. 2.1, no simple correction scheme can be adequate in case of high wall/flow temperature differences and a numerical solution should be sought (Kays & Crawford (1980)).

The core of engineering knowledge on the topic has been built on flat plate or pipe flows, and a lack of appreciation of the magnitude of this effect on NGV flows seems to exist. The available experimental data on the topic are very limited. Those available are restricted to 2D midspan results and of limited accuracy to draw any significant conclusions Dunn et al. (1984) performed an experimental analysis on the effect of T_w on HTC for a turbine blade. Their results suggest an effect of TR on Nu but, due to experimental configuration, they have different external Re for the three different TR and one of the three analysed cases has different freestream turbulence intensity levels than the other two. The experimental survey by Andrews & Bradley (1957) only contains midspan average Nu from cascade tests but no detailed local data are available. Finally also Hylton et al. (1983) discussed the issue of HTC dependence on TR using the C3X vane and report a general trend of increase of HTC levels with the reduction of the temperature ratio. However the experimental uncertainties were so high, over 20% at the trailing edge, that the reported effect of T_w was within the experimental uncertainty for most of the blade.

This brings to a few interrelated questions that are going to be addressed in the first part of this thesis:

1. What is the effect of wall temperature on HTC for a typical NGV flow?
2. Is there an observable feedback of the thermal field in the aerodynamics?
3. Are there regions, both in 2D and 3D flow conditions, where the effect of wall temperature is higher?
4. Can a traditional power-law scaling of Nu account for this dependence?

5. If not, is it possible to develop a new heat transfer scaling method that could account for it?

Given the answers to these questions, and the impact that CHT effects have in turbomachinery flows, as discussed in Sec. 2.2, it becomes fundamental to include Conjugate Heat Transfer in the design process. For this reason in the remaining part of this thesis the following arguments will be discussed:

1. Implementation of CHT capabilities in Rolls Royce CFD solver Hydra,
2. Validation of the implemented solver,
3. Conjugate investigation of a double-wall trailing edge cooling system.

Part II

Investigation of the interaction between heat transfer and aerodynamics

Chapter 3

Methods and validations

Investigation of the connection between aerodynamics and heat transfer in high pressure turbines has been carried out using ANSYS Fluent second order coupled pressure-based solver (Chorin (1968)). The MT1 nozzle guide vane (Chana et al. (2001)) geometry has been taken as test case.

The opening section of the chapter introduces the issue of heat flux non linearity with T_w and its implication on HTC definition. The baseline methodology for HTC calculation is presented and the relative numerical sensitivities are discussed.

The rest of the chapter is dedicated to illustrate the numerical modeling and the necessary validation and numerical consistency checks performed.

During the course of this thesis work, numerical simulations have been carried out first on a 2D geometry, to be later extended to the full 3D MT1 NGV. Though very simple in content and conclusions, both 2D and 3D validation and mesh dependency will be discussed hereafter for the sake of completeness. This is necessary as results from both 2D and 3D simulations will be discussed in the following chapters.

3.1 Baseline two-point HTC calculation procedure

Scaling of the wall convective heat flux to obtain HTC can be achieved through different ways. The absence of a unique definition for the heat transfer coefficient is itself source of uncertainties in turbine heat transfer predictions. Typically the use of a reference temperature allows to scale wall heat flux using:

$$HTC = \frac{\dot{q}}{(T_{ref} - T_w)} \quad (3.1)$$

Using Eqn. (3.1) it is possible to calculate HTC using a single CFD solution for a specified wall temperature. The reference temperature T_{ref} is often taken as T_{0in} .

Despite its wide use, this definition for HTC can result in inconsistent results, particularly non-physical values for HTC. For instance, we might need to work out HTC at a near adiabatic condition. This need can be due to experimental constraints or can also be caused by large local variations in fluid driving temperatures either at a high speed flow with an appreciable recovery effect, or due to a non-uniform inlet temperature traverse. Equation (3.1) may then return a zero (or even negative) HTC, as the temperature difference in the denominator does not reflect the true local driving temperature difference. Given that HTC is the reciprocal of the equivalent thermal resistance, a zero or negative HTC would be physically meaningless. This scenario would also violate the basic physical consideration that heat transfer is driven by a temperature difference, thus a zero heat flux can only result from a zero temperature difference and HTC cannot be zero. Even from a practical perspective, a zero HTC would make the scaling in Eqn. (3.1) non-workable.

It is also frequent to use the adiabatic wall temperature T_{aw} to calculate the local heat transfer coefficient. The use of T_{aw} as driving temperature is certainly more 'sound' than using a constant T_{ref} as described above. The temperature difference driving the heat transfer is calculated based on the adiabatic conditions, where one

would expect the heat flux to be zero.

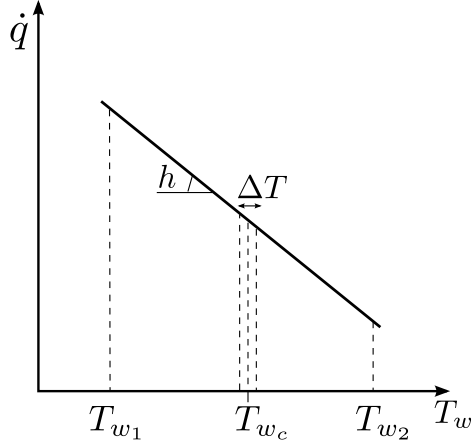
The described approaches both suffer, from the preconception that heat transfer descriptors should be invariant with the wall temperature. That is, more generally, the main inconsistency behind Eqn. (3.1): the implicit assumption that both HTC and T_{ref} are aerodynamically-determined and, as such, are not dependent on heat transfer phenomena. This becomes questionable in compressible flows (including low speed flows in presence of considerable temperature gradients).

Since the issue of the dependence of HTC with the wall temperature is the main purpose of this investigation the HTC definition as given in Eqn. (3.1) is not pursued here. A two-temperature method as a finite difference approximation of the local tangent of a $\dot{q} - T_w$ curve has been used instead.

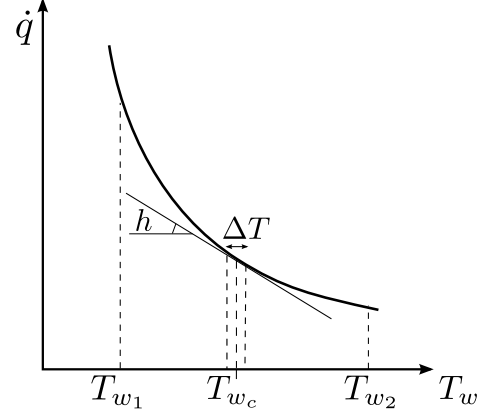
The approach is better explained with the help of Fig. 3.1. Consider a heat transfer problem, in which we need to work out HTC for a range of wall temperatures between T_{w1} and T_{w2} . Assuming that heat transfer descriptors are uniquely aerodynamically-determined, there should then be, for a given blade configuration and flow conditions, two unknowns (h , T_{ref}) in Eqn. (3.1) both invariant with thermal boundary conditions. It then follows that these two unknowns are independent of the wall temperature. This scenario corresponds to the widely held conventional wisdom that the heat flux varies linearly with the wall temperature (Fig. 3.1(a)). As such, the slope (h) can be easily worked by taking a two-point method. Two CFD calculations at two specified wall temperatures are carried out. Defining T_{w+} and T_{w-} as the two wall temperatures chosen and $\dot{q}_+$ and $\dot{q}_-$ the corresponding heat transfer distributions over the blade surface h is then obtained by means of Eqn. (3.2).

$$h = \frac{\dot{q}_+ - \dot{q}_-}{T_{w+} - T_{w-}} = \frac{\Delta \dot{q}}{\Delta T} \quad (3.2)$$

When $\dot{q}$ is a linear function of T_w , as commonly assumed, (Fig. 3.1(a)) the slope of the



(a) Linear $\dot{q} = f(T_w)$: slope (h) is independent on T_{w_c} and ΔT



(b) Non-linear $\dot{q} = f(T_w)$: slope (h) is dependent on T_{w_c} and on ΔT

Figure 3.1: Schematics of the differences between linear and non-linear $\dot{q} = f(T_w)$ relation when the h value is of interest.

curve (i.e. the HTC) is the same for any T_w . In this condition the calculated h does not depend on either T_w or ΔT . Also the local driving temperature, independent of T_w as well, would be the conventionally used T_{aw} . In this respect it is worth highlighting how the two-point method of Eqn. (3.2) should be seen as a generalization of the classical heat flux scaling of Eqn. (3.1), rather than an alternative definition.

In the case of a non-linear heat flux relation, the working mode for HTC calculation needs to be local to T_w . Noting that Eqn. (3.2) can be interpreted as a finite difference approximation of the differential $\frac{d\dot{q}}{dT_w}$, the local HTC for a given wall temperature T_w can be worked out by a local finite difference approximation of the local tangent to the heat flux curve at T_w . The key point here is that, for this non-linear case, the validity range of the two points method shrinks at the limit to a point: only an infinitesimal ΔT is correctly representing the local slope of the curve, which is now depending on T_w .

3.2 Numerical method

3.2.1 Solution method

Fluent uses a Finite Volume, cell-centered discretization. The solver used for Fluent calculations has been the steady pressure-based coupled solver. In pressure based solvers a pressure equation is derived from the continuity and momentum equations, allowing to obtain a velocity field that is satisfying the conservation of mass. In the coupled solver the equations are solved simultaneously.

Second order upwind spatial discretization has been used, where values of flow variables at cell faces are reconstructed through a Taylor series expansion of the cell-centered solution. Gradients have been calculated using the Least Squares Cell-Based method.

The discretized equations result in a linear system of equations. This system is solved using a Gauss-Seidel method. Acceleration of the solver is achieved through the use of an Algebraic Multi Grid (AMG) method. Stabilization of the solver is obtained through variable relaxation.

For more information on the details of the numerical implementation of the used schemes the reader is referred to Ansys (2011).

3.2.2 Fluid model

For all the calculations performed with Fluent, air has been used, considered as a perfect gas ($R = 287.2 [J/(kg K)]$). Viscosity has been modeled with Sutherlands' law and the variation of thermal conductivity with temperature has been accounted for by using a piecewise linear relation. The isobaric specific heat was taken constant ($C_p = 1005.2 [J/(kg K)]$). The specified fluid properties resulted in a Prandtl number, based on inlet conditions, $Pr \approx 0.7$ for the flat plate cases and $Pr \approx 0.9$ for the MT1 cases. A value of the turbulent Prandtl number $Pr_t = 0.85$ has been assumed

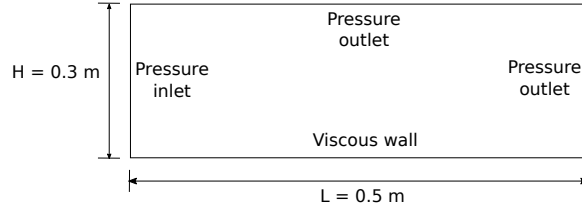


Figure 3.2: Domain used for flat plate calculation with type of boundaries used.

p_{0in} @ $Re = 5 \times 10^4$	101326.5	[Pa]
p_{0in} @ $Re = 1 \times 10^6$	101906.2	[Pa]
p_{sOut}	101325.0	[Pa]
T_{0in}	300	[K]
T_w	310	[K]
Turbulence intensity	1%	[-]
Turbulence length scale	1×10^{-3}	[m]

Table 3.1: Summary of the CFD boundary conditions used for flat plate validation.

for all simulations, except where explicitly indicated otherwise.

3.2.3 Computational domains and boundary conditions

3.2.3.1 Flat plate studies

The flat plate calculations have been performed on the computational domain of Fig. 3.2. Both top and right boundaries of the domain were specified as pressure outlets. This allowed to reduce the disturbances to the boundary layer evolution due to the truncation of the domain.

The boundary conditions used are summarized in Tab. 3.1.

3.2.3.2 MT1 studies

Both 2D and 3D computational domains used for the MT1 calculations are showed in Fig. 3.3. The 3D computations use the same kind of boundaries as the 2D domain, with the addition of the hub and shroud endwalls, both treated as viscous walls.

Boundary conditions used, matching experimental conditions, are summarized in

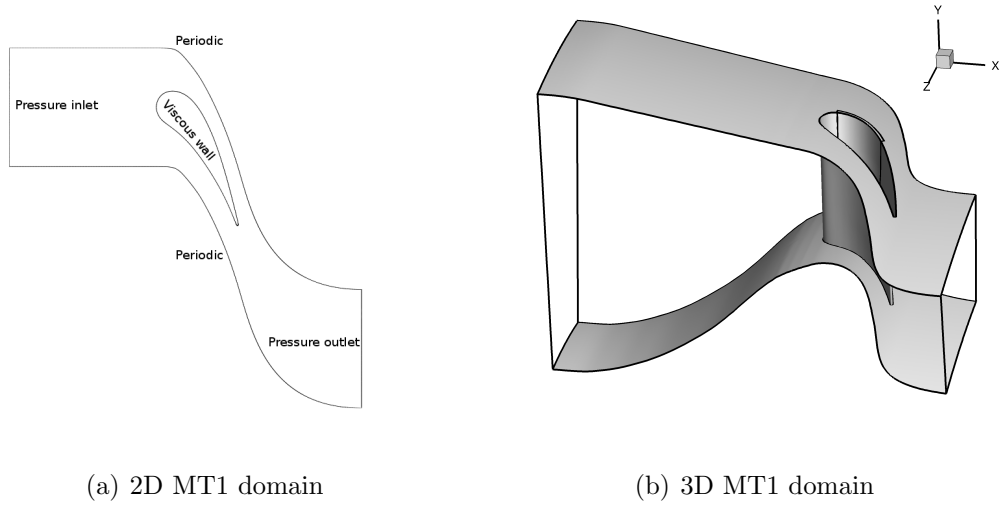


Figure 3.3: Domains used for MT1 calculations with type of boundaries used.

p_{0in}	460000.0	[Pa]
M_{isOut}	0.98	[-]
T_{0in}	444.0	[K]
T_w	288.0	[K]
Turbulence intensity	5% at leading edge	[-]
Turbulence length scale	5.0×10^{-3}	[m]

Table 3.2: Summary of the CFD boundary conditions used for 2D MT1 validation.

Tab. 3.2 for the 2D calculations and in Tab. 3.3 for 3D calculations. The blade wall and endwalls (for the 3D case) have been modeled as isothermal at ambient temperature being the test rig a short duration facility. For three-dimensional calculations a radial distribution of static pressure at the NGV outlet has been prescribed, based on spanwise-averaged mixing-plane calculations on the same geometry by Rahim et al. (2013).

3.3 Flat plate heat transfer validation

Heat transfer validations checks have been performed first on simple incompressible boundary layer flow, taken as a reference case because of the wide amount of well

p_{0in}	460000.0	[Pa]
M_{isOut}	1.05 <i>hub</i> – 0.91 <i>tip</i>	[-]
T_{0in}	444.0	[K]
T_w	288.0	[K]
Turbulence intensity	6% <i>at leading edge</i>	[-]
Turbulence length scale	5.0×10^{-3}	[m]

Table 3.3: Summary of the CFD boundary conditions used for validation tests.

established correlations and analytical solutions to compare with. Both laminar and turbulent regimes have been considered.

Reynolds numbers analyzed are 5×10^3 , 5×10^4 , 1×10^5 for the laminar cases and 5×10^5 and 1×10^6 for the turbulent ones. The difference between the temperature of the external flow and the plate temperature was of 10 K for all the cases. Maximum free-stream Mach number was about 0.1 for the highest Reynolds number case. The domain (Fig.3.2) has been discretized with around 1×10^5 nodes. Wall y^+ was less than unity for the whole length of the plate and for all the Reynolds numbers covered. Both Spalart-Allmaras (Spalart & Allmaras (1992)) and $k - \omega$ SST (Menter (1994)) turbulence models have been compared. Laminar CFD results have been compared with the analytical solution obtained by Kays & Crawford (1980) of Eqn. (3.3).

$$Nu_x = 0.332 Pr^{1/3} Re_x^{1/2} \quad (3.3)$$

The turbulent CFD predictions have been compared with the correlation by Reynolds et al. (1958), shown in Eqn. (3.4)

$$Nu_x = 0.0296 Pr^{0.6} Re_x^{0.8} \left(\frac{T_w}{T_\infty} \right)^{-0.4} \quad (3.4)$$

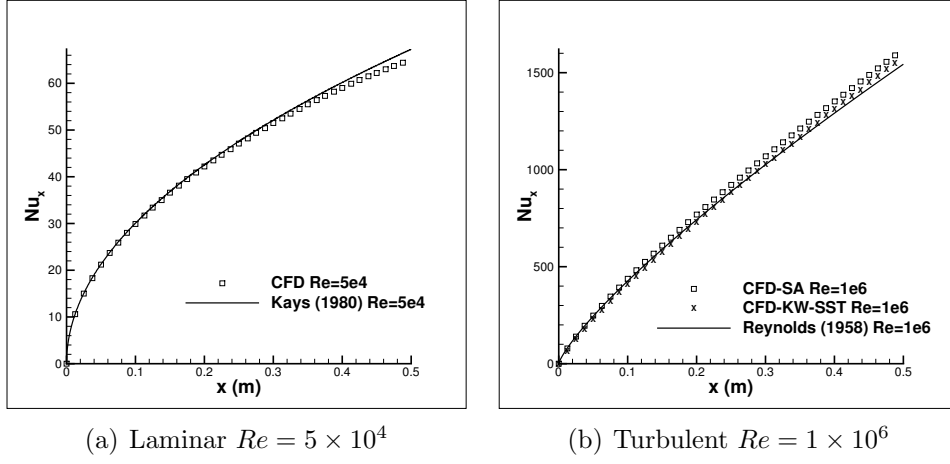


Figure 3.4: Calculations on laminar and turbulent boundary layer for constant T_w boundary condition: Nu comparison between Fluent and the reference correlation.

Nusselt number for flat plate calculations has been calculated as from Eqn. (3.5)

$$Nu_x = \frac{\dot{q}x}{(T_\infty - T_w)k} \quad (3.5)$$

consistent with the way it is defined in the correlations used as a benchmark.

For the sake of brevity only the results for Reynolds numbers 5×10^4 (laminar) 1×10^6 (turbulent) are shown in Fig. 3.4. The other analyzed cases show similar accuracy. Good agreement for all cases, either laminar or turbulent, has been found. No marked differences between the two turbulence models has been observed. A mesh dependency study has been also performed on the flat plate validations by changing the number of nodes in the direction normal to the plate in the boundary layer blocks (extending for 1/10 of the total height H of the domain). The mesh used in the preceding calculations had 50 nodes; it has been compared with a refined mesh having 70 nodes but the same wall y^+ . The study, the results of which are not shown here for brevity, confirmed the mesh independence of the results of Fig. 3.4.

3.4 MT1 Validation

In this section the results of solver validation against MT1 NGV experimental data obtained by QinetiQ Chana et al. (2001) will be illustrated. As mentioned at the beginning of the chapter, both 2D (at midspan) and 3D computations have been carried out on the MT1 nozzle guide vane. For this reason the validation tests on both 2D and 3D domains will be illustrated in the following paragraphs.

Either the 2D and the 3D domains (Fig. 3.3) have been discretized using only quadrilateral/hexahedral elements. Boundary layer resolution has been achieved using a O-grid next to the blade wall. A low Reynolds approach has been used for the laminar sublayer resolution: no wall functions have been used throughout the work.

The transitional behavior of the boundary layer on the blade suction side, showing frontal laminar region followed by a transition to turbulent boundary layer, has been captured by turning off the turbulence production terms in a defined near-wall region. Such region is highlighted in red in Fig. 3.5 and Fig. 3.8(a). This is equivalent to a forced trip transition at the end of this region. The transition point and the extension of the laminar region have been adjusted to match the experimental results. This approach to model transition is still widely used as noted by Denton (2010), and allowed not to consider, at first, the influence of wall temperature on transition point. This issue, and its implications, are analyzed separately and will be discussed in Ch. 4.

Turbulence modeling on the both geometries has been accomplished by using both Spalart-Allmaras (Spalart & Allmaras (1992)) and $k - \omega$ SST (Menter (1994)). In this section only the results obtained using Spalart-Allmaras turbulence model will be discussed. The results of the turbulence model sensitivity study will be presented in Sec. 3.5.3.

Aerodynamic validation has been performed by comparing the predicted M_{is} with midspan data available from experiments. For the thermal validation experimental data of non-dimensional wall heat flux q/q_{ref} at midspan have been used. The defini-

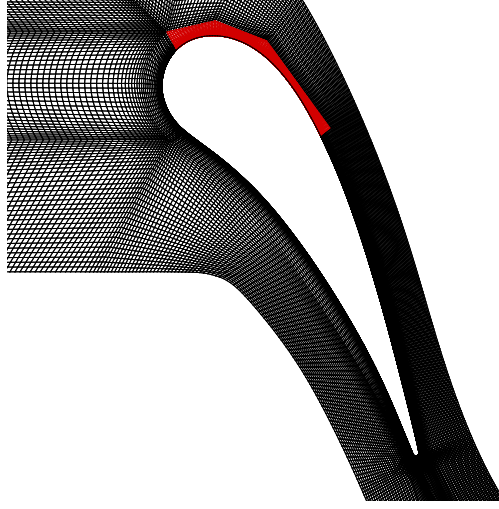


Figure 3.5: Grid used for the 2D calculations. Highlighted in red is the forced laminar flow region.

tion of the non-dimensional wall heat flux is given in Eqn. (3.6) where C is the blade chord and k is the thermal conductivity of air evaluated at the inlet temperature.

$$\dot{q}/\dot{q}_{ref} = \frac{\dot{q}C}{(T_{0in} - T_w)k} \quad (3.6)$$

3.4.1 2D MT1 validation

The profile used for the 2D validation is the midspan section of the MT1 NGV. The mesh used had around 5×10^4 nodes. Wall distance of the first row of nodes has been chosen so to have $y^+ < 1$ on the whole blade surface. A detail of the grid used can be shown in Fig. 3.5.

Comparison of the isentropic Mach number M_{is} between CFD and experiments is shown in Fig. 3.6. The rear part of the profile shows bigger discrepancies with experiments compared to the front part. This is however not due to modeling problems but rather to the fact that the full 3D geometry has a converging meridional channel (shown in Fig.3.3(b)) that a 2D domain cannot account for. In fact the results of the

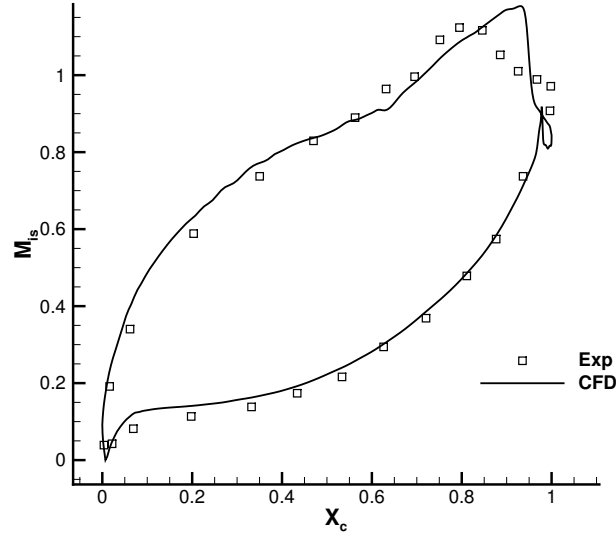


Figure 3.6: Aerodynamic validation for the 3D MT1 domain

aerodynamic validation on the midspan section of the full 3D geometry (Fig. 3.9) do show a much better accuracy on the rear part of the profile using the same solver and boundary conditions.

Figure 3.7 shows the results of the thermal validation. As was the case for the aerodynamic validation some discrepancies can be observed between experimental data to CFD results. Those differences are comparable to the ones found in other CFD studies on the same geometry (Khanal et al. (2013), Lad et al. (2012)), and are reasonably good in the context of the case considered.

The discrepancies observed may be caused by experimental uncertainties and uncertainties in the experimental boundary conditions. Particularly no mention is given in Chana et al. (2001) about the former. It also worth pointing out that the experimental data are obtained in a from a 3D, transient, 1.5 stage experimental facility.

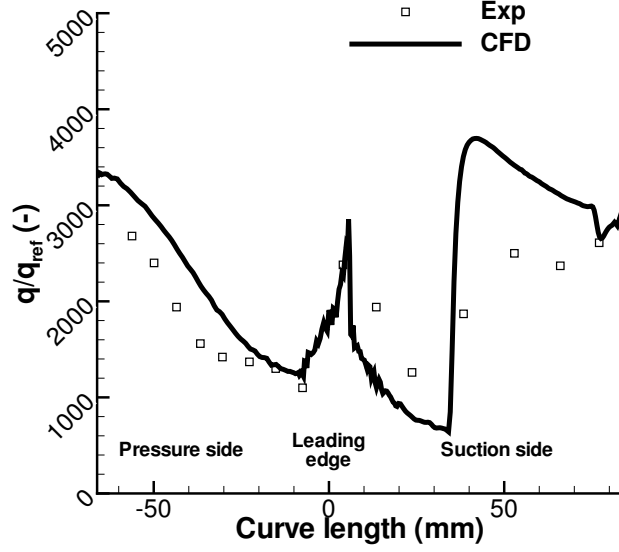


Figure 3.7: Heat transfer validation for the 2D MT1 domain

3.4.2 3D MT1 validation

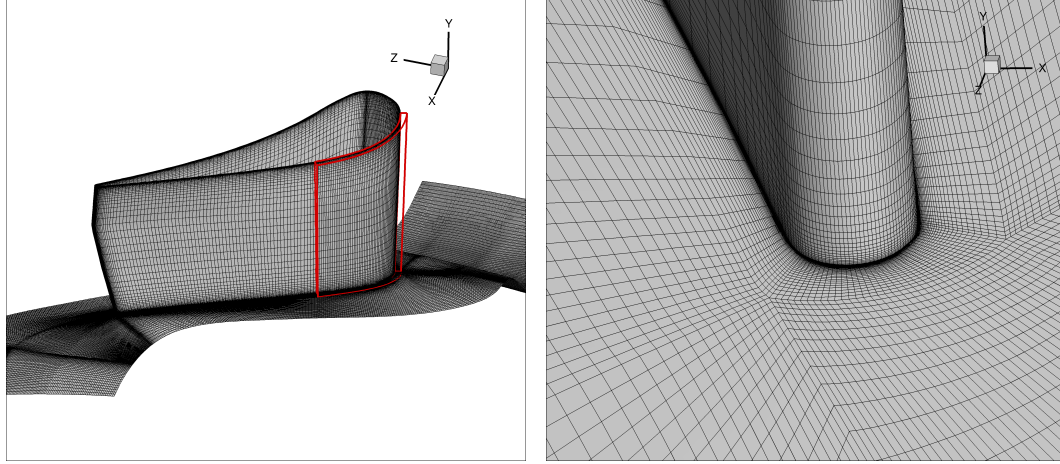
Three-dimensional calculations have been performed on the domain shown in Fig. 3.3(b). The final mesh used counts approximately 4 million nodes. A global view of the mesh and a close up of the trailing edge at the hub are shown in Fig. 3.8 . In Fig. 3.8(a) is highlighted in red the laminar region used for tripping boundary layer transition.

Results of the aerodynamic validation, comparing M_{is} experimental data CFD predictions, are shown in Fig. 3.9 . Excellent agreement with experimental data can be observed.

The heat transfer validation results are shown in Fig. 3.10 and show to be comparable to the validation results of the 2D configuration previously discussed. As such the same conclusions drawn in Sec. 3.4.1 hold.

3.5 Further numerical sensitivity studies

In this section the results of the necessary numerical consistency checks will be discussed. The results of the grid dependency studies for the 2D and the 3D MT1 will



(a) Computational grid: laminar region highlighted

(b) Grid detail: trailing edge-hub

Figure 3.8: Grid used for the performed 3D calculations.

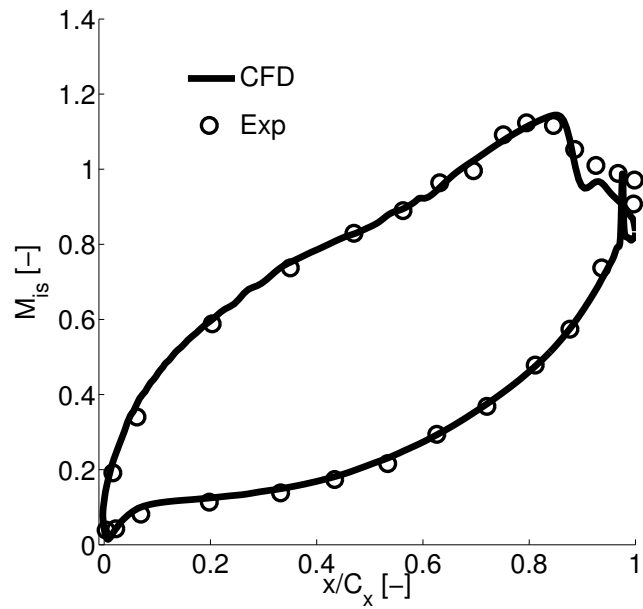


Figure 3.9: Aerodynamic validation for the 3D MT1 domain.

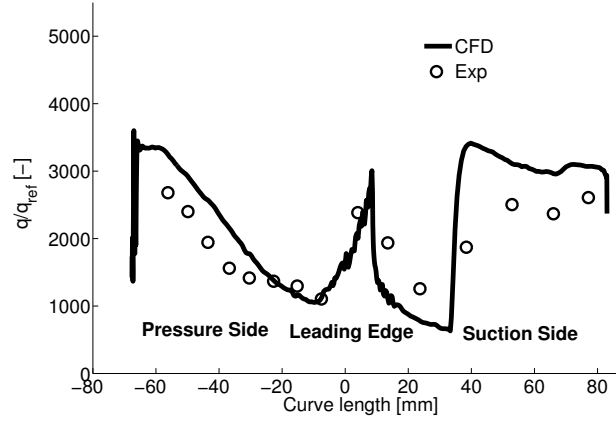


Figure 3.10: Heat transfer validation for the 3D MT1 domain

be presented as well as the turbulence model sensitivities.

Finally the numerical sensitivities of the two-point method for HTC calculation introduced in Sec. 3.1 will be discussed.

3.5.1 2D MT1 mesh dependency

The mesh density in the blade passage has been considered adequate for the considered case. The mesh dependency study on the 2D MT1 has been accomplished thus by altering the number of nodes in the boundary layer O-mesh surrounding the blade profile. The final mesh used, shown in Fig. 3.5 had 30 nodes across the boundary layer mesh. Three different grids with 20, 30 and 40 nodes respectively have been compared. In the mesh dependency study, wall HTC (calculated according to Eqn. (3.2)) distributions obtained using the three different grids have been compared. Results of the mesh dependency study are summarized in Fig. 3.11.

The mesh with 20 nodes in the O-mesh blocks has noticeably different predictions in some zones on the pressure side. Negligible differences between the 30 and 40 nodes grids has been found, indicating grid independent results for the mesh with 30 boundary layer nodes that has been actually used in further analyses.

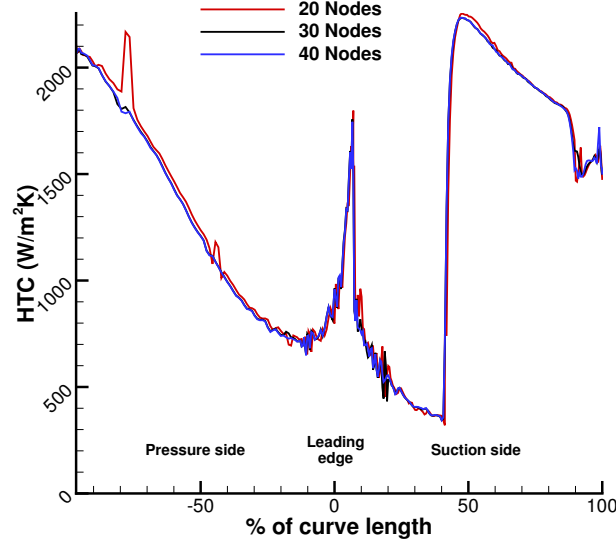


Figure 3.11: Mesh dependency study for the 2D case: HTC distributions for the 0.6 TR case with different BL nodes count.

3.5.2 3D MT1 mesh dependency

A grid independency study has been performed also on the 3D domain. The node count has been varied in both span-wise and pitch-wise directions across the whole passage. During all mesh tests the near wall distance has not been varied as it was deemed necessary to keep $y^+ < 5$ to ensure a proper near-wall resolution.

The results of the mesh dependency study for heat transfer calculations are shown in Fig. 3.12. Two mesh densities (4 and 5 million nodes) have been compared. The results indicate grid independency for the analyzed case: the mesh with 4 million nodes has been used for the following 3D calculations.

3.5.3 MT1 turbulence model sensitivity

To rule out the influence of the turbulence model on the results shown so far, a turbulence model sensitivity study has been carried out both on the 2D and the 3D MT1 simulations.

The results of the study are summarized in Fig. 3.13. The predictive performance

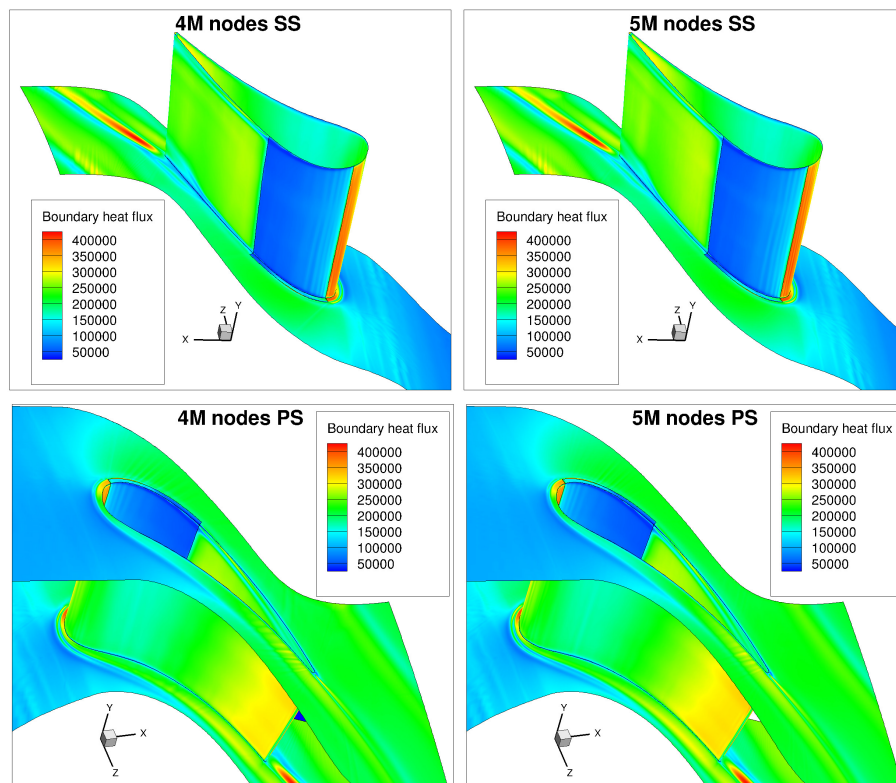


Figure 3.12: Mesh dependency study. Boundary heat flux (W/m^2) for the two mesh densities.

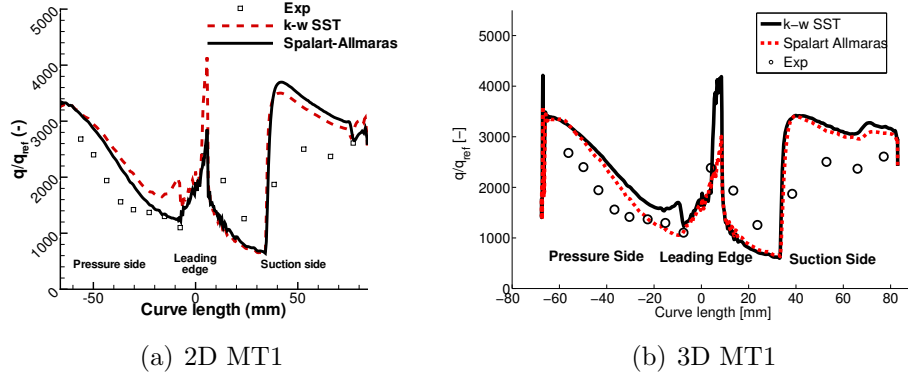


Figure 3.13: Turbulence model sensitivity study for both 2D and 3D MT1.

of both Spalart-Allmaras (Spalart & Allmaras (1992)) and $k-\omega$ SST (Menter (1994)) turbulence models has been assessed. The two turbulence models do not show marked differences. The only notable mismatch between the two can be seen at the leading edge, where the $k-\omega$ SST shows higher values of the heat flux. This is consistent with the known behavior of two equations models to produce high levels of turbulence in regions of high flow gradients with respect to one equation models.

The two equations $k-\omega$ SST model has been used for the analyses carried out on the 3D geometry, while the 2D midspan MT1 results have been mainly obtained using Spalart-Allmaras¹.

3.5.4 HTC ΔT sensitivity

As introduced in Sec. 3.1, the method used to calculate the wall HTC, can be seen as a finite differencing operator approximating the local slope of the heat transfer curve slope. The final numerical check that has been performed concerns the sensitivity of the two-point HTC calculation (Eqn. (3.2)) with the temperature difference ΔT between the two analyzed cases.

¹Obtained HTC distributions have been checked to be independent of the turbulence model and a second set of CFD calculations has been performed also using $k-\omega$ SST. The results of this study are discussed in Sec. 4.2

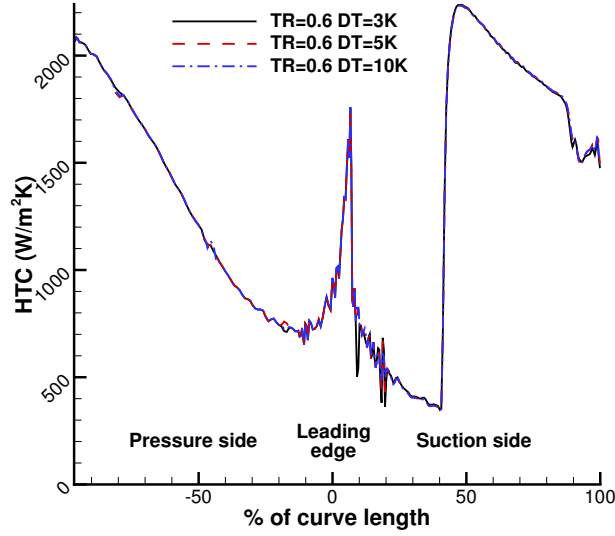


Figure 3.14: HTC distributions for $(T_{w+} - T_{w-})$ of 3 K, 5 K and 10 K.

From a numerical view point, use of a too big ΔT will lead to large truncation errors, whilst a too small ΔT may lead to unacceptable rounding-off errors. In order to consistently calculate the HTC distribution on the surface and take into account the possible variability of h with T_w , the two points method of Eqn. (3.2) has been used with a $\Delta T = T_{w+} - T_{w-} = 5 \text{ K}$ small enough to consider $\dot{q} = f(T_w)$ to be linear within a reasonable approximation in that temperature interval. Numerical studies have been carried out to confirm that no effect of ΔT was observed on the obtained values of HTC. The results, shown in Fig. 3.14 confirm that.

In the next chapter tests at several Temperature Ratios (TR) $T_w = TR \cdot T_\infty$ will be presented. For each TR two calculations have been run with constant wall temperatures $T_{w+} = T_w + 2.5 \text{ K}$ and $T_{w-} = T_w - 2.5 \text{ K}$. The obtained values of h have been considered to be representing correctly the HTC levels at T_w considering the small ΔT considered.

3.6 Final remarks

The numerical methods of the solver used for this first part of the thesis have been briefly introduced.

Thermal validation of the solver has been performed against flat plate boundary layer correlations as well as experimental data on the MT1 NGV. The results of the validation tests can be considered satisfactory and are in line with the validation cases available in open literature on the same case.

The two point method as a finite-difference approximation for HTC calculation has been introduced and the relevant numerical sensitivities with respect to the used temperature difference have been discussed.

Once assessed the reliability of the numerical modeling of the solver on the analyzed case, the MT1 geometry will be used in the next chapters to study the impact of the wall thermal boundary condition on HTC and aerodynamics.

Chapter 4

Wall temperature effect on HTC and aerodynamics

In this chapter a deeper analysis of the link between aerodynamics and heat transfer for typical NGV flow is carried out.

As discussed in Ch. 3, both 2D midspan and 3D simulations have been performed on the MT1 NGV. The first part of the chapter will be focused on midspan data, the discussion will then be extended to fully 3D flows.

The effect of wall temperature on HTC will be analyzed and the connection between aerodynamics and heat transfer will be explored, highlighting the impact that heat transfer can have on the boundary layer profiles as well as on shocks and secondary flow structures.

In the last section the use of transition models will be introduced in spite of the tripped transition approach (see Sec. 3.4.1) and a discussion on the implications of transition point dependence with the wall temperature is presented.

4.1 Boundary conditions

To analyze the effect of wall temperature on HTC, the relevant non-dimensional parameter is the Temperature Ratio (TR) defined as:

$$TR = \frac{T_w}{T_{0in}} \quad (4.1)$$

The boundary conditions used in all simulations have been kept the same as the ones used in the validation study (see Tab. 3.2 for the 2D case studies and Tab. 3.3 for the 3D cases), exception made for the blade wall temperature, changed from time to time to match the desired temperature ratio.

HTC distribution on the blade wall is obtained using the two-points method introduced in Sec. 3.1 (see Eqn. (3.2)), so each HTC distribution presented refers to two CFD computations at two different wall temperatures (T_{w+} and T_{w-}) apart by a $\Delta T = T_{w+} - T_{w-} = 5 \text{ K}$ (see Sec. 3.5.4). A summary of wall temperatures used for the temperature ratios discussed in the present chapter is presented in Tab. 4.1. All calculations consider $T_{0in} = 444 \text{ K}$.

$TR [-]$	$T_{w+} [K]$	$T_{w-} [K]$
0.6	269	264
0.7	313.5	308.5
0.75	335.5	330.5
0.8	357.5	352.5
0.95	424.5	419.5
0.99	442	437

Table 4.1: Summary of the wall temperatures used for the analyzed temperature ratios.

4.2 2D case studies

4.2.1 HTC dependence with temperature ratio

The influence of TR on external HTC has been quantified by comparing simulations at three different temperature ratios: 0.95 (close to adiabatic conditions), 0.75 and 0.6 (highly cooled blade). The results are shown in Fig. 4.1 from which the influence of wall temperature on HTC stands out very clearly. Differences in HTC values at trailing edge between lowest and highest TR analyzed are as big as 25% towards the blade trailing edge.

Interestingly, it can also be observed that in the stagnation point region the effect of TR is less pronounced than in the zones close to the trailing edge. This indicates that the change in flow properties in the near-wall region, due to the different thermal boundary condition, alters the evolution of the boundary layer. This statement is further supported by the boundary layer traversing data discussed in Sec. 4.2.3. The key point to be made here, and it is of fundamental importance for the discussion that will follow, is that it is not only the local wall temperature that influences the HTC but also the entire upstream history of the boundary layer subject to the cooled surface. At the leading edge, where the boundary layer has just been formed, the effect of wall temperature is minimum, but it increases considerably as the boundary layer develops over the blade surface. The progressive accumulation inside the boundary layer of fluid with a different temperature with respect to external flow increases the influence of wall temperature as the boundary layer itself develops.

The sudden separation of the HTC curves on the suction side, after the transition point, is likely to be caused by the point-trip transition used. The abrupt turbulent transition may, in fact, cause overshooting in the prediction of heat transfer levels after transition. This however should not impact the general considerations drawn on the effect of TR on HTC. This statement is proved by the results of the conjugate

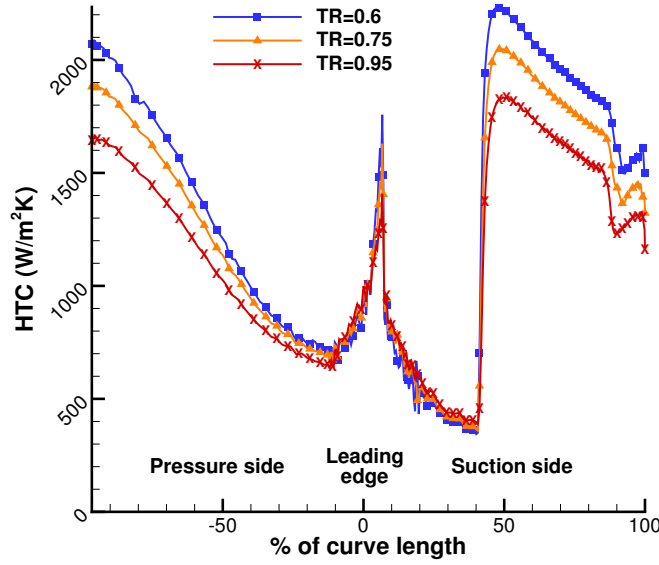


Figure 4.1: Dependency of external HTC distribution on TR for 2D midspan calculations.

study of Sec. 4.2.1.1. Heat transfer in the metal here helps reducing the HTC peak after transition but the same HTC vs TR dependency can be observed. Also similar effect of TR is observed on both pressure and suction side, though the former is not interested by transition.

The results shown in Fig. 4.1 have been obtained using the Spalart-Allmaras turbulence model and a turbulent Prandtl number of 0.85. Tests have been performed using also a reduced Pr_t of 0.4 (Fig. 4.2) and using the $k - \omega$ SST Menter (1994) turbulence model (Fig. 4.3). Both those studies confirmed the conclusions drawn beforehand. The results shown in the rest of this section on 2D midspan data are all obtained using Spalart-Allmaras turbulence model.

As stated in Sec. 3.1, a change of the HTC with the wall temperature indicates a non-linearity of Newton's law of cooling. From the results discussed before the analyzed case shows a higher degree of linearity at the leading edge, where TR dependency of HTC is less pronounced. The non-linearity increases when moving towards the trailing edge, with increased differences in HTC with T_w . Figure 4.4 compares the heat transfer levels at different wall temperatures on two positions: one near the

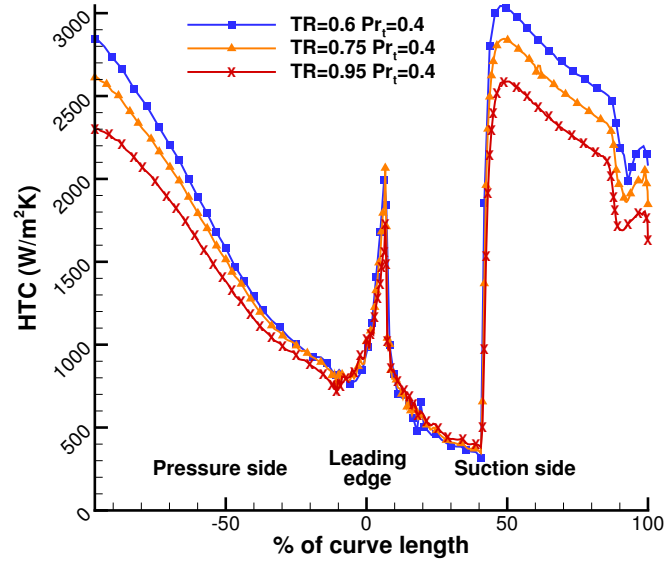


Figure 4.2: Dependency of external HTC distribution on TR using a Pr_t of 0.4 for 2D midspan calculations.

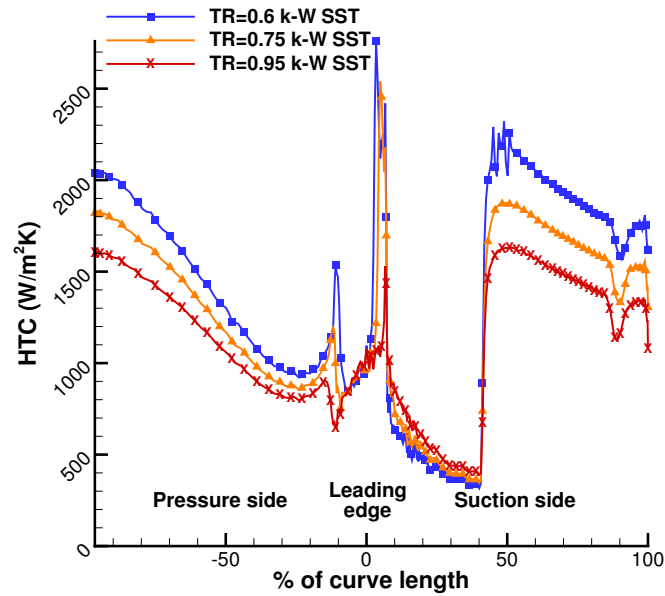


Figure 4.3: Dependency of external HTC distribution on TR using $k-\omega$ SST turbulence model Menter (1994) for 2D midspan calculations.

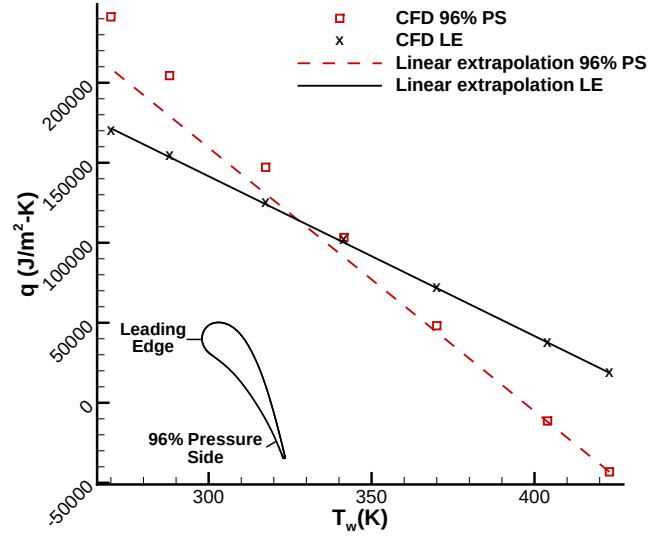


Figure 4.4: Calculated $\dot{q} = f(T_w)$ compared with linear approximation for the two wall points considered.

leading edge and the other near the pressure side trailing edge. The lines are linear extrapolations (not a linear fit of the data) of the heat flux using the HTC value close to the adiabatic condition ($TR=0.95$). The increased non-linearity for the point on the pressure surface near the trailing edge is evident with respect to the leading edge point. The results of Fig. 4.4 show that the use of a HTC obtained at a high temperature ratio can introduce significant error when scaled to a lower temperature ratio in regions where non-linear effects are not negligible.

4.2.1.1 Further examination with CHT solution

The results previously shown have been obtained considering only external aerodynamics. A conjugate study has been performed to confirm the results obtained previously when the wall is treated as an interface rather than a boundary. In this study, the interface temperature between the solid and the fluid is not specified anymore and it is also not constant. The conjugate study is important to confirm and validate the previously shown non-conjugate results and to highlight the limitations

of the uncoupled approach to calculate external HTC distributions.

The blade has been equipped with four internal cooling channels. For each of them a value of HTC and a bulk fluid temperature have been imposed. A more detailed discussion of the internal cooling boundary conditions is presented later in this section. The blade solid mesh is shown in Fig. 4.5: solid thickness has been chosen aiming to obtain a uniform wall temperature. The metal has been assumed to be ASTM310 with a thermal conductivity variable with temperature. Considering the average levels of HTC and material properties, the resulting average Biot number is around 0.1. Calculation of h has been done again using Eqn. (3.2). Perturbation of the wall temperature in the conjugate case has been accomplished by varying by 5 K the temperature of the coolant in the cooling channels.

In this section a comparison between the results previously shown, obtained using external CFD, and the h distributions obtained from the conjugate calculations is provided. In order to make a sensible comparison between the conjugate and the non-conjugate cases, the CHT simulations have been performed aiming to obtain a blade wall temperature as uniform as possible and corresponding to the same TR as the non-conjugate case. Due to the differences in external HTC from pressure side to suction side, it has been impossible to obtain the same level of T_w on both sides of the profile without changing the boundary conditions for the cooling holes. For this reason for a given TR, the h distributions for the conjugate case have been obtained with two simulations, each using two different sets of cooling boundary conditions. The internal cooling boundary conditions used for the Pressure Side (PS) and Suction Side (SS) simulations for the different temperature ratios are grouped in Tab. 4.2.

In all the conjugate cases the mean wall temperatures on both suction side and pressure side have been matched with the corresponding T_w on non-conjugate simulations within 2K. The variability of the wall temperature increased as the temperature ratio decreased and was also bigger on the suction side than on the pressure side due



Figure 4.5: Mesh used for the conjugate solution.

		TR = 0.95				TR = 0.75				TR = 0.6			
PS		T_{b1}	T_{b2}	T_{b3}	T_{b4}	T_{b1}	T_{b2}	T_{b3}	T_{b4}	T_{b1}	T_{b2}	T_{b3}	T_{b4}
		413	414	417	418	300	306	296	302	217	210	190	180
		h_1	h_2	h_3	h_4	h_1	h_2	h_3	h_4	h_1	h_2	h_3	h_4
		4000	3500	3500	3500	4000	3500	3500	3500	4000	3500	3500	3500
SS		T_{b1}	T_{b2}	T_{b3}	T_{b4}	T_{b1}	T_{b2}	T_{b3}	T_{b4}	T_{b1}	T_{b2}	T_{b3}	T_{b4}
		419	442	450	430	300	306	296	302	216	196	196	195
		h_1	h_2	h_3	h_4	h_1	h_2	h_3	h_4	h_1	h_2	h_3	h_4
		4000	3500	3500	3500	4000	3500	3500	3500	4000	3500	3500	3500

Table 4.2: Summary of the internal cooling boundary conditions. Coolant bulk temperature T_b values are given in K , HTC h values in $W/(m^2K)$. Subscripts refer to the cooling hole number from LE to TE.

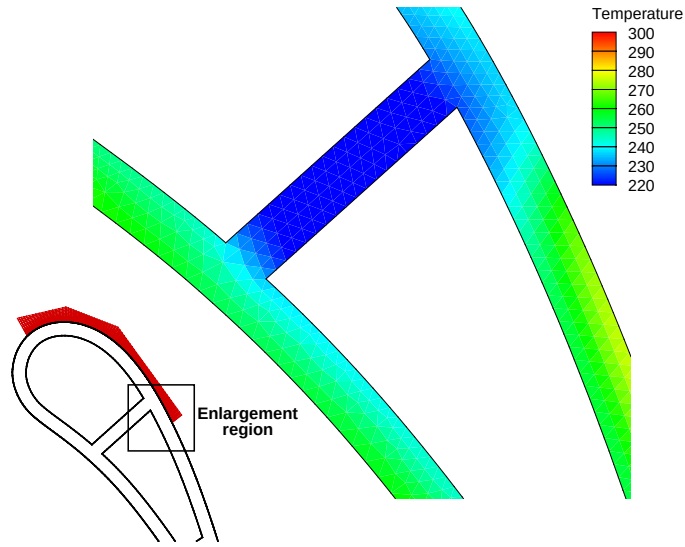


Figure 4.6: Detail of metal temperature field for the 0.6 TR case.

to the steep flow acceleration and the sudden change of the external HTC around the turbulent transition. Figure 4.6 highlights the variations of the wall temperature on the suction side region around boundary layer transition. The importance of such wall temperature non-uniformity on the suction side region, after transition, becomes clear when looking at the results of Fig. 4.7. The picture compares the HTC distributions at different temperature ratios for both conjugate and non-conjugate cases. Globally a really good agreement is found and the conjugate results confirm fully the effects of wall temperature on HTC obtained with the uncoupled study. However the region on the suction side after transition presents the highest differences between the conjugate and non-conjugate calculations. Such differences are to be ascribed to the temperature non-uniformities on the blade wall in that region, i.e. to the marked difference in wall boundary conditions between the conjugate and non-conjugate cases. For those regions with relatively smooth and small temperature variations, the wall temperature between the conjugate and non-conjugate cases are better matched and the heat transfer coefficients from the two solutions show a really good agreement. The solid part on the suction surface after the transition is subject to a rapid change

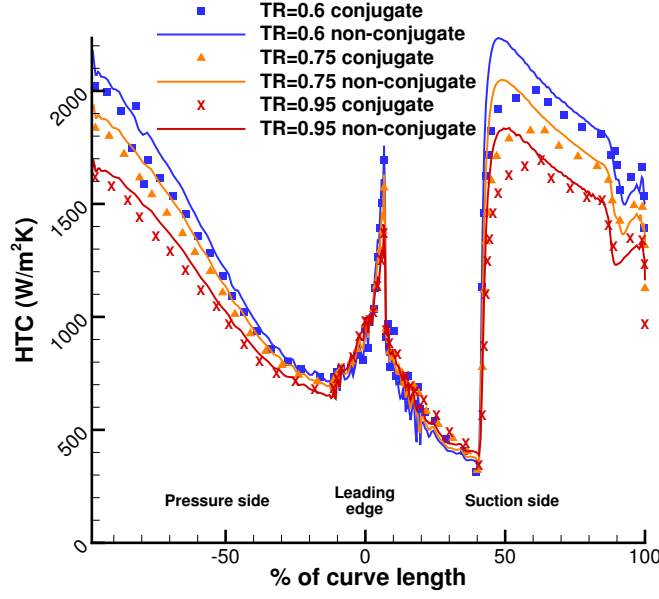


Figure 4.7: Dependency of external HTC distribution on TR: comparison between conjugate and non-conjugate cases.

in heat flux, that the designed cooling system cannot absorb. This causes the local surface temperature and the HTC to change. This discrepancy in local HTC due to the difference in the corresponding local T_w in fact does reinforce the central point that HTC does depend on the wall temperature.

4.2.2 Fluid driving temperature variation with TR

The effect of TR on the adiabatic wall temperature has been also studied for the 2D midspan case. Once obtained the value of h for each temperature ratio, the value of the corresponding T_{aw} has been determined using the following relation:

$$T_{aw} = T_w + \frac{\dot{q}}{h} \quad (4.2)$$

The results of the study are shown in Fig. 4.8. The quantity on the ordinate axis $\frac{T_{aw} - T_{awTR=0.95}}{T_{awTR=0.95}}$ expresses the relative difference between the calculated fluid driving temperature T_{aw} and the driving temperature for the 0.95 TR case. This quantity

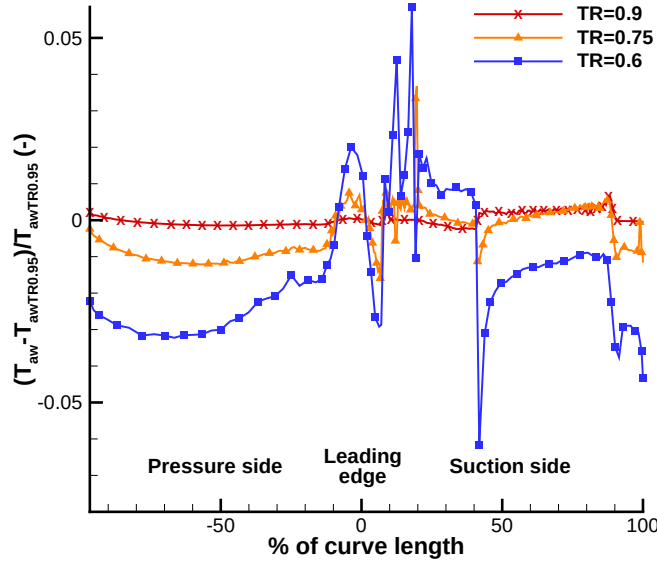


Figure 4.8: Dependency of T_{aw} distribution on TR for the 2D midspan results.

is indicative of the relative differences in T_{aw} for different temperature ratios. The results indicate that TR does influence the fluid driving temperature as well, though to a lesser extent than it does on HTC. Also, as for the HTC plots, differences look minor near the leading edge than at the trailing edge.

4.2.3 Thermal response of boundary layer profile

In order to help to gain some insight into the effects of wall temperature on external aerodynamics, a detailed examination of the boundary layer has been carried out based on the uncoupled results of Sec. 4.2.1. Boundary layer behavior is, in fact, fundamental when convective heat transfer is of concern. Traversing of the the boundary layer in different sections around the blade profile has been performed. This demonstrated that temperature, density and momentum profiles inside the boundary layer are profoundly affected by wall temperature. The characteristics of flow inside the boundary layer are strongly dependent on density, velocity and viscosity as they influence momentum transport and wall gradients. The combined influence of the temperature distribution in the boundary layer on density, velocity and viscosity, has

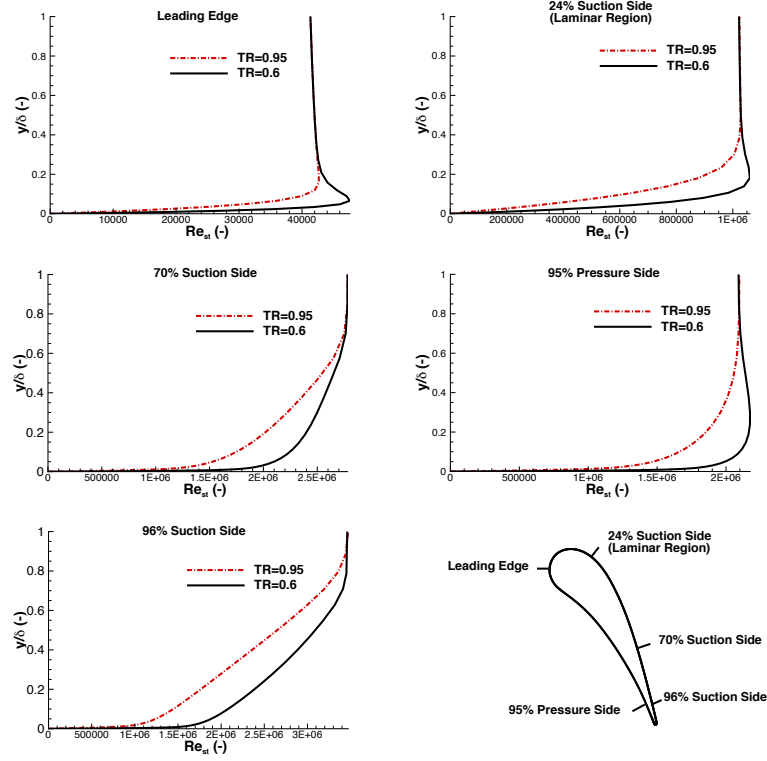


Figure 4.9: Re_{st} distributions across boundary layer at 5 traversing sections for 0.95 and 0.6 TR cases.

been examined defining a local streamwise Reynolds number $Re_{st}(y)$ as:

$$Re_{st}(y) = \frac{u(y)\rho(y)x}{\mu(y)} \quad (4.3)$$

where y is the local coordinate normal to the blade surface, u is the local velocity magnitude, ρ is the fluid density, μ its viscosity and x is the stream-wise distance of the point where the traversing is made from the stagnation point. This definition of Re_{st} is a Reynolds number but it is taken here simply as a non-dimensional parameter to illustrate the impact of wall temperature on aero-thermal development of boundary layer. Traversing has been made in five locations on the blade, the positions of which are given in Fig. 4.9. The figure presents the comparison of Re_{st} distributions between the 0.95 and 0.6 TR cases for all the traversing sections. In the figure the y coordinate has been non-dimensionalized by the local boundary layer thickness δ .

The results of Fig. 4.1 support the idea that the change of wall temperature alters the aerodynamic behavior of the boundary layer. Moreover the change in TR does not reflect identically along the profile but differences seem to amplify as the boundary layer develops. In the regions where the boundary layer is just formed (i.e. around leading edge), discrepancies in the Re_{st} distributions between adiabatic and cooled case are spatially limited in the zone of the boundary layer that is closer to the wall. On the other hand, approaching the trailing edge, the difference in Re_{st} extends to nearly the whole boundary layer thickness. The progressive extension of the differences in Re_{st} to outer zones of the boundary layer signifies also a progressive modification of the external flow as the boundary layer develops further downstream on the blade surface. Overall, the Re_{st} development in the boundary layer traverses highlights the importance of the upstream history effect.

The different development of the aero-thermal field along the blade surface modifies also the turbulent quantities ¹. This is, in broad terms, easily understandable as the turbulence model itself represents a connection between the flow solution and turbulent quantities (ultimately turbulent viscosity).

Figure 4.10 shows the traversing for the turbulent viscosity at the leading edge and at 95% pressure side. The positions of the traversing stations are identical to the ones reported in Fig. 4.9. It is clear the effect of the different wall temperature on the evolution of turbulent viscosity. The case with lower wall temperature shows, at the late pressure side, higher levels of μ_t . The two distributions are pretty much identical instead when the boundary layer has just formed. The different sensitivity of the laminar and turbulent regions with respect to wall temperature suggests that the effect of the different evolution of the boundary layer on the turbulent quantities plays a prominent part in the local dependence of HTC with wall temperature.

¹The particular link between the flow solution and turbulent quantities depends, naturally, on the choice of turbulence model. A deeper discussion of the mechanisms of this connection for different turbulence models is outside the scope of this thesis.

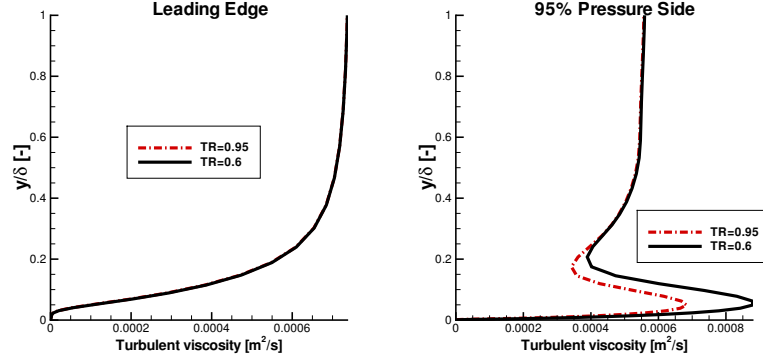


Figure 4.10: Traversing of turbulent viscosity μ_t distributions across boundary layer at leading edge and 95% pressure side for 0.95 and 0.6 TR cases.

The effect of a different development of the boundary layer along the profile should not be overlooked as it translates in a different momentum thickness at the trailing edge, and that is known to be a major driver for profile losses (see Stewart (1955)).

4.2.4 Influence of T_w on trailing edge flow shocks

Another investigation has been carried out to check if the change in the boundary layer properties due to the wall temperature could have a noticeable influence on the trailing edge shock pattern of the blade, particularly considering that the trailing edge is the region where the effect of 'flow history' would be felt more. This is relevant to the applicability of the conventional wisdom assuming that external flow should not be influenced by the wall surface heat transfer. As suggested by Lovely & Haines (1999) the shock detection variable defined in Eqn. (4.4) has been used to determine the exact position of the shock waves.

$$M_n = \vec{M} \cdot \vec{\nabla} p \quad (4.4)$$

In the formula $\vec{M}$ is the Mach number vector defined by:

$$\vec{M} = M \frac{\vec{U}}{\|\vec{U}\|} \quad (4.5)$$

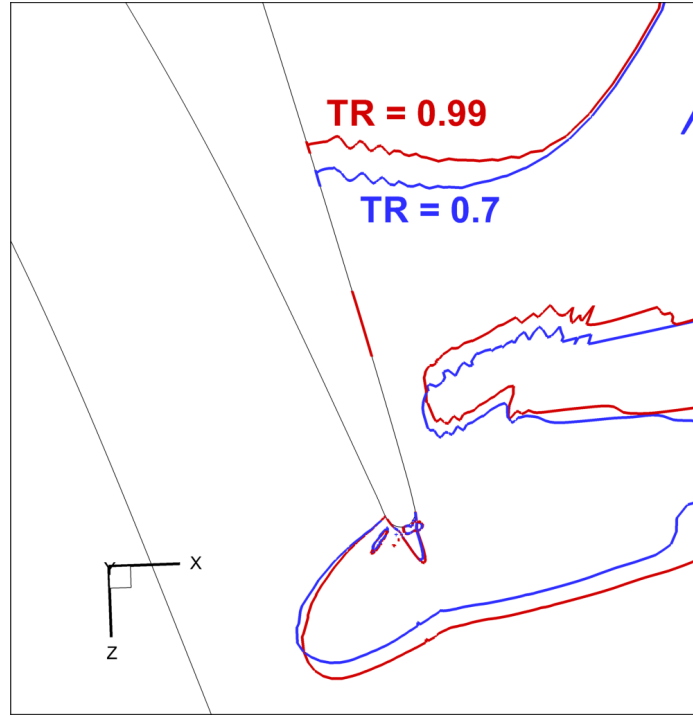


Figure 4.11: Suction side shock position for the quasi-adiabatic (TR=0.99) and cooled (TR=0.7) case.

where $\vec{U}$ is the velocity vector. M_n indicates the value of the Mach number normal to a shock wave. Knowing that M_n is always equal to unity across a shock wave it is possible to use it as a shock detection variable. Figure 4.11 shows the flood lines of $M_n = 1$ in the trailing edge region for both temperature ratios analyzed. A downstream shift of the suction side shock for the cooled case is observed. The shift distance is of the same order of magnitude as the trailing edge thickness. The pressure side shock does not seem to be affected in this case by the wall temperature.

The observed shifting of the trailing edge shock reinforces concept of the existing coupling between thermal boundary condition and external aerodynamics. The small change in the position of the shock, found for a relatively old blade design, should not be underestimated. It can be expected that, in modern transonic blades, characterized by stronger shock waves and reflections, the wall temperature might have a stronger influence on the whole shock pattern and, potentially, on blade profile losses. The associated changes in boundary blockage around the trailing edge region may well

also have a non-negligible impact on the passage flow capacity. The simulations were fully converged and the shock position was not varying with the iterations.

4.3 3D case studies

4.3.1 HTC dependence with temperature ratio

With a similar approach to the one described for the 2D midspan section, analysis of the dependence of HTC with TR has been performed also on the 3D domain of Fig. 3.8. Two temperature ratios have been analyzed in the 3D case: $TR = 0.99$ (quasi-adiabatic) and $TR = 0.7$ (cooled case).

Figures 4.12, 4.13, 4.15 and 4.14 show the comparison of HTC contours for the nearly adiabatic and the cooled cases on blade pressure side, suction side, hub and shroud endwalls respectively. The HTC percent difference between the two cases is calculated as:

$$\frac{HTC_{TR=0.7} - HTC_{TR=0.99}}{HTC_{TR=0.99}} 100 \quad (4.6)$$

The midspan section of the 3D NGV showed a dependency of HTC with TR consistent with the results presented in Sec. 4.2.1, as expected, with lower HTC differences at the stagnation point than at the trailing edge. The main focus of the 3D simulations, however, has been the investigation of the effect of T_w on the HTC distributions on the entire blade and endwalls surfaces.

Differently from the pressure side (Fig. 4.12), the suction side (Fig. 4.13) clearly shows the signature of secondary flows on heat transfer distributions for both temperature ratios. From Fig. 4.13 it can be seen that suction side-endwall regions, dominated by secondary flows, do show the highest differences in HTC (up to 40%). Similar HTC difference levels can be seen in the 50% span region, where the flow field

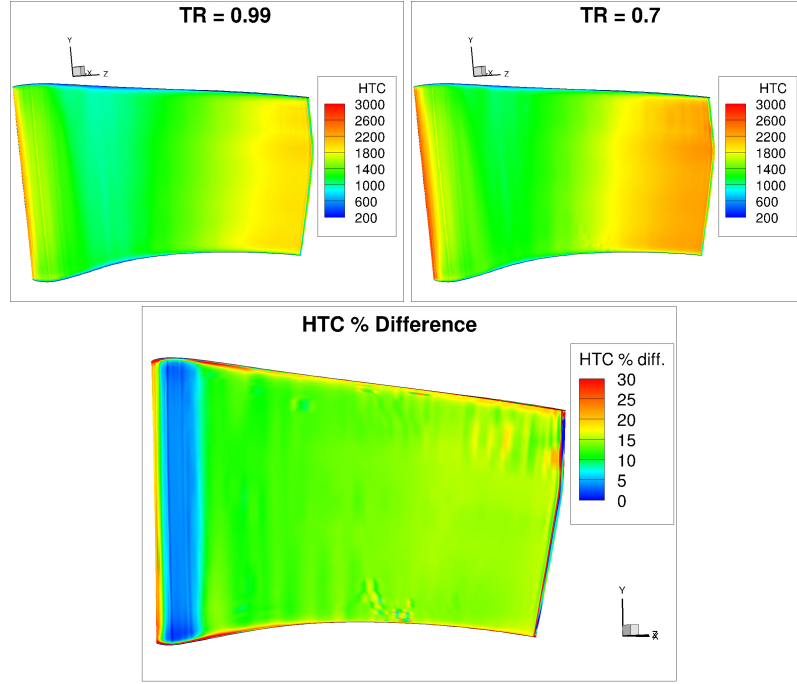


Figure 4.12: HTC dependency on TR, on the vane pressure side.

is largely 2D. Here the differences can be explained by the influence of wall temperature on the position of the suction side shock². Similar observations can be made by comparing the endwalls HTC contours of Fig. 4.14 and Fig. 4.15. The highest differences in HTC levels are concentrated in the regions where secondary flows are dominant. These results reinforce the evidence that a link between aerodynamics and heat transfer does exist, and HTC changes with wall temperature cannot be ascribed only to the changes in gas properties. For this reason any attempt to correct this T_w dependence of HTC, without taking into account local flow history, cannot be considered effective in predicting the correct HTC levels.

Also the strong dependence of HTC on TR in regions characterized by highly 3D flows, makes inapplicable any boundary layer based correction (that could still be sought when considering a purely two-dimensional flow). These observations are fundamental for the discussion on the scaling of HTC with wall temperature that will

²Shifting of the suction side shock has been observed also on the 3D midspan section using the same methodology used in Sec. 4.2.4. Results of this analysis are not presented here for the sake of brevity.

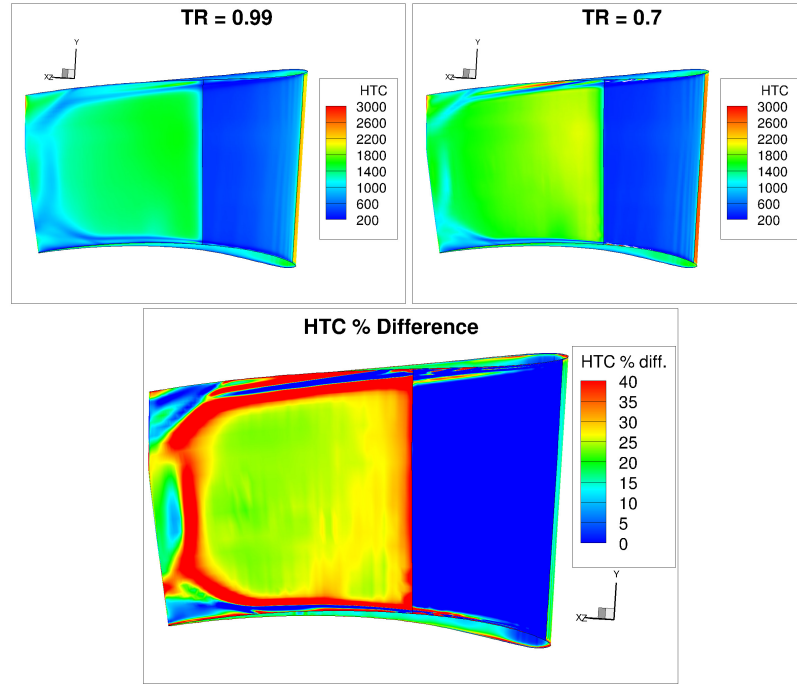


Figure 4.13: HTC dependency on TR, on the vane suction side.

presented in more detail in Ch. 5.

4.3.2 Influence of TR on secondary flows

A further examination of the effect of wall temperature on secondary flows has been carried out. Figure 4.16 shows flow streamlines at the suction side-endwalls corner. Streamlines are consistently generated from the same points in the two cases. Some differences are appreciable between the quasi-adiabatic and cooled case with a more pronounced streamline convergence towards the midspan for the higher TR case.

A close look at the changes in secondary flows has been possible also by visualizing the direction of wall shear stresses in the form of wall streamlines. The results are shown in Fig. 4.17. Differences in the wall shear stress direction due to different developments of secondary flow structures with wall temperature are noticeable. In particular, Fig. 4.17 suggests that the radial inward movement on the rear blade suction surface near the casing shroud for 0.99 temperature ratio is seemingly stronger

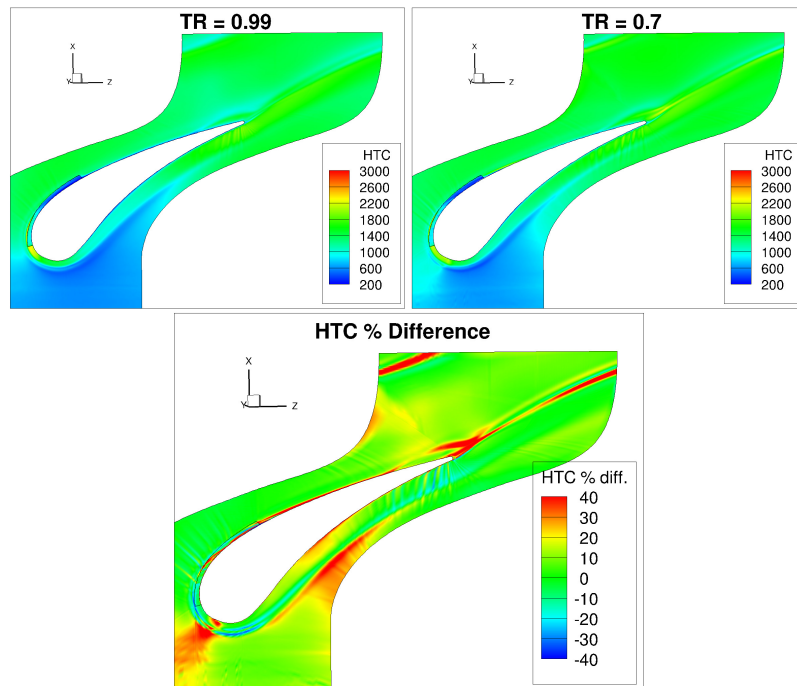


Figure 4.14: HTC dependency on TR, on the shroud endwall.

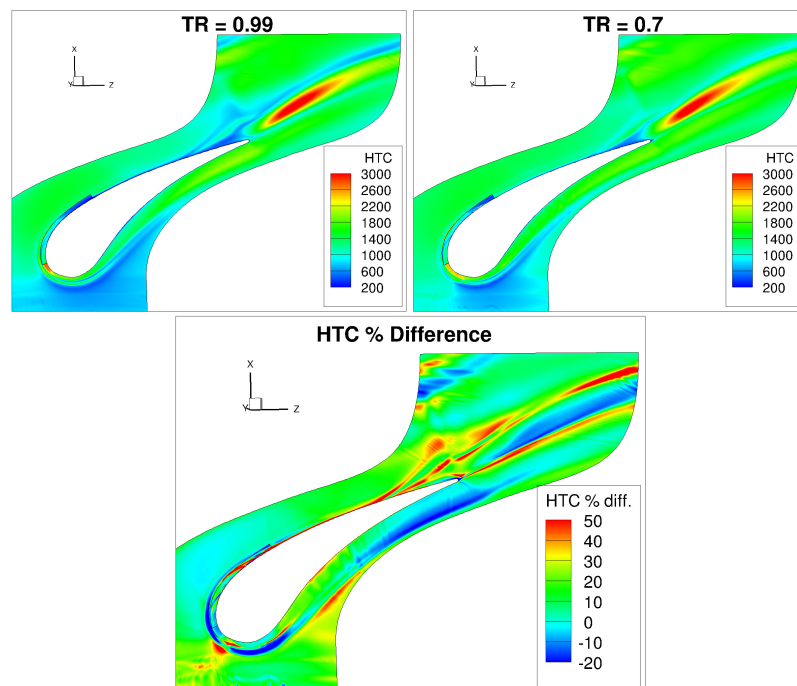


Figure 4.15: HTC dependency on TR, on the hub endwall.

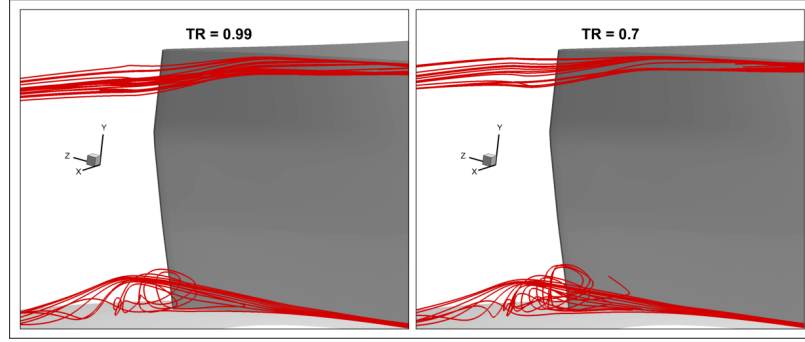


Figure 4.16: Streamlines at suction side-endwalls corners for quasi-adiabatic (TR=0.99) and cooled (TR=0.7) case.

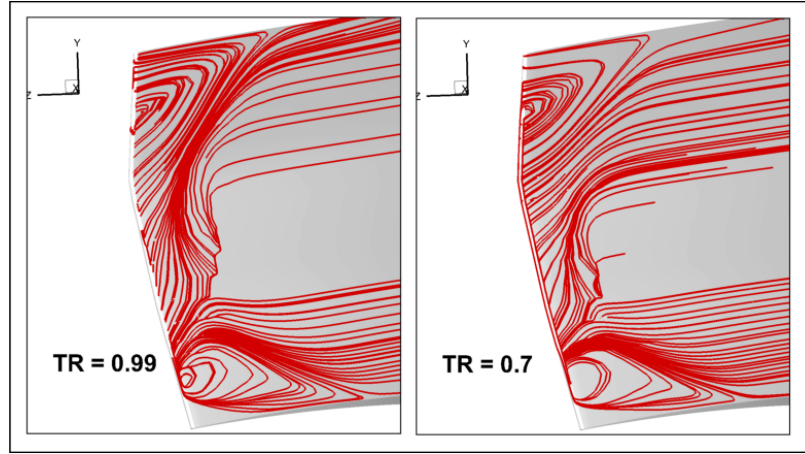


Figure 4.17: Wall shear stress direction on suction side for quasi-adiabatic (TR=0.99) and cooled (TR=0.7) case.

than that for 0.7. This might be attributed to the higher flow density due to the cooling (hence the higher local fluid inertia, and resistance to the secondary flow transport) in in the near-wall region of the cooled case.

Also a comparison of the contour plots of wall shear stress between the quasi-adiabatic and cooled case is reported in Fig. 4.18. It is possible to see how the wall temperature impacts not only the position of the limiting streamlines but also the levels of shear stress in the trailing edge region. This again suggests the presence of a direct coupling between aerodynamics and heat transfer.

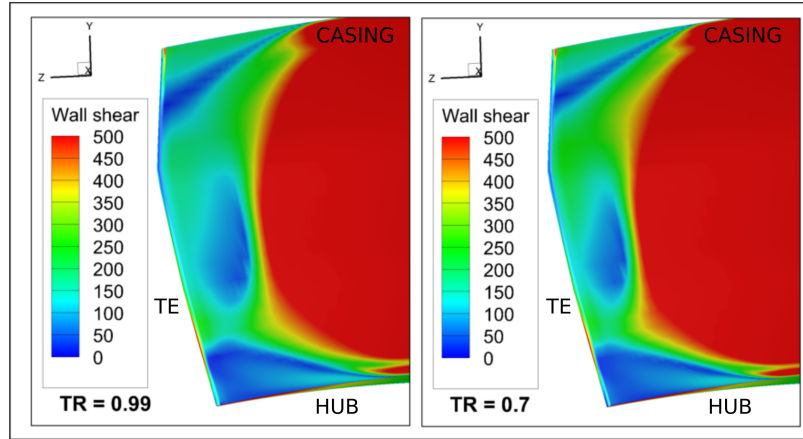


Figure 4.18: Wall shear stress magnitude on the late part of the suction side for quasi-adiabatic ($TR=0.99$) and cooled ($TR=0.7$) case.

4.3.3 Influence of TR on passage mass flow

The changes of flow density in the blade passage due to the wall temperature have been examined also. The results are shown in Fig. 4.19. The plots show the density contours at a cut plane positioned after the trailing edge. The position of the cut plane is also shown. A first remark that should be done is that the cut plane is not normal to the rotational periodicity axis so the results of Fig. 4.19 do not show a periodic match despite the CFD solution being periodic.

Sensible differences can be observed comparing the two density fields. A remarkable observation is the fact that such differences are not confined to the wall regions but penetrate to a higher extent in the whole passage due to convection of boundary layer material ascribed to secondary flows. In general it can be seen how the highest differences in density can be observed in the regions where most of the aerodynamic losses are generated (near-wall regions, trailing edge wake, secondary flows). As such this effect should not be overlooked in the aero design process.

Considering the differences in flow density as well as secondary flows across the blade passage, the influence of wall temperature on the overall blade passage capacity has been also assessed. All the CFD runs examined had mass flow converged to less than 0.1%. The 0.7 TR case showed a 0.5% increase in mass flow with respect to

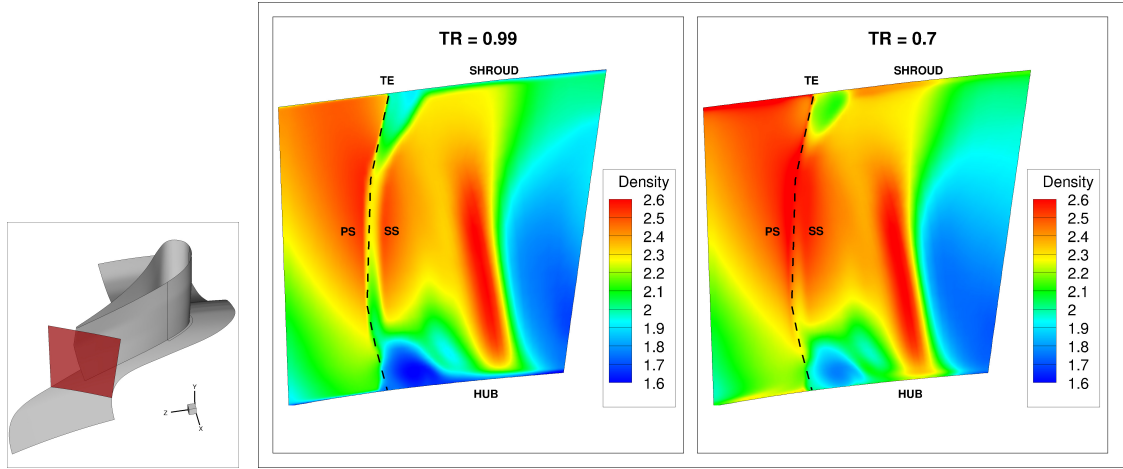


Figure 4.19: Density contours at TE cut plane for quasi-adiabatic (TR=0.99) and cooled (TR=0.7) case.

the quasi adiabatic case. Considering that the general trends of blading performance increases nowadays are aimed at fractions of percent points, the impact of such a change in blade capacity should thus not be overlooked, also in light that temperature ratios lower than 0.7 can be typical of heavily cooled NGVs.

4.4 Influence of wall temperature on transition

The fixed transition treatment as described above, though simple, does not allow to capture the effect of wall temperature on the transition location. The results obtained in this section use transition models in which the transition location is determined as part of the solution, allowing to further explore the effects of TR on external aerodynamics as well as the implications of the transition point movement on the HTC definition.

Due to the high mesh density required by transition models, the analysis has been carried out on the 2D mid-span section of the MT1 NGV. A refined grid with 150k nodes has been used for the study. Wall y^+ has been kept lower than one. Calculations have been performed using two well-established transition models available in Fluent:

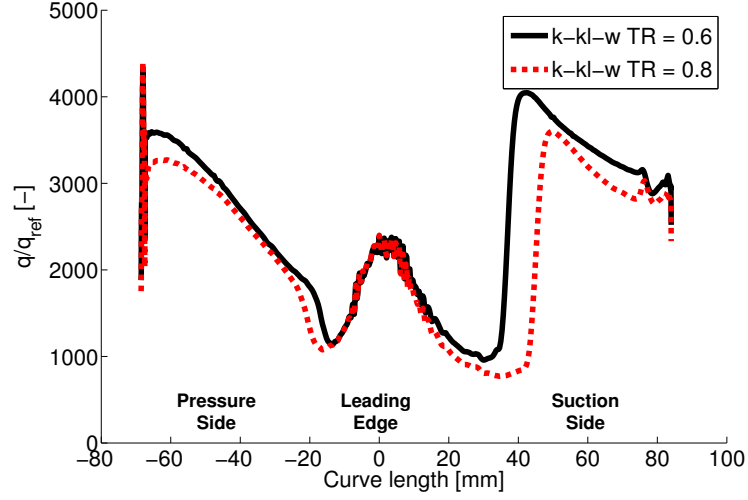


Figure 4.20: $\dot{q}/\dot{q}_{ref}$ (Eqn. (3.6)) for TR=0.8 and 0.6 using the $k - kl - \omega$ transition model.

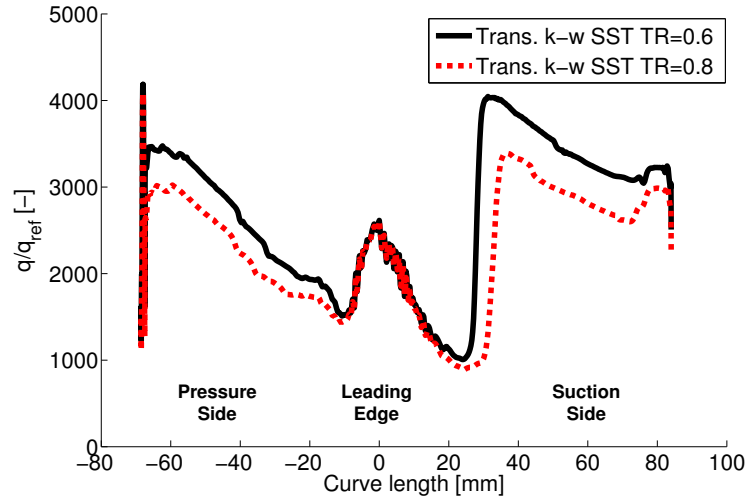


Figure 4.21: $\dot{q}/\dot{q}_{ref}$ (Eqn. (3.6)) for TR=0.8 and 0.6 using the transitional SST model.

transitional SST (Menter et al. (2006)) and $k - kl - \omega$ (Walters & Cokljat (2008)).

Figures 4.20 and 4.21 show wall non-dimensional heat flux (Eqn. (3.6)) obtained using the $k - kl - \omega$ and the transitional SST respectively. Despite some offset in the predicted transition point and length, both transition models are consistent in predicting an early onset of transition due to the cooling.

The reasons of this anticipated transition should be sought in the local alterations of the boundary layer flow due to wall cooling. Figure 4.22 shows traversing of the

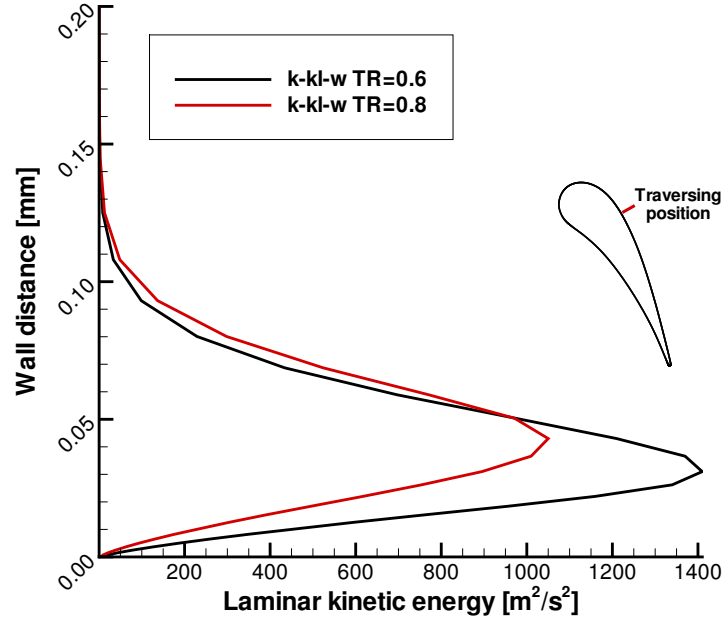


Figure 4.22: Boundary layer traversing of laminar kinetic energy before the transition point for TR=0.8 and 0.6 using the $k - kl - \omega$ transition model.

boundary layer prior to transition point using $k - kl - \omega$ transition model. The plot represents the distribution of laminar kinetic energy in the boundary layer. This quantity is used in the $k - kl - \omega$ transition model to model pre-transitional boundary layer disturbances (see Walters & Cokljat (2008)). Plots of Fig. 4.22 clearly show that the alterations of the aero-thermal field in the boundary layer results in higher levels of laminar kinetic energy for the lower temperature ratio case, promoting an earlier transition.

This effect of cooling in promoting boundary layer transition as predicted presently is qualitatively in line with that observed experimentally by Back et al. (1969) and Kercher et al. (1982) and is also consistent with the destabilizing effect of cooling on bypass transition described by Reshotko & Tumin (2004). A quantitative evaluation of the accuracy of the used transition models is beyond the scope of this work. This observed behavior in the present calculation is interesting as a cooled boundary layer is often perceived as being more stable (thus should accordingly have a delayed

transition instead). Given the consistency in the response to wall temperature shown by the two transition models used, the following analysis will be carried out by only using $k - kl - \omega$ transition model.

Given the movement of the transition point and the drastic change of the heat flux from the laminar to turbulent regimes, a key question of interest is: how does this movement affect the HTC and its working procedure? The issue should be seen now both in terms of accuracy and robustness. The latter is particularly relevant in the context of stability of CFD-FEA loosely coupled methods using HTC as a buffer parameter.

In Fig. 4.23, the HTC distributions for TR=0.6 is shown for $k - kl - \omega$ transition model. HTC is here calculated using the conventional 2-point method with $\Delta T = 10K$ (Eqn. (3.2)). The movement of the transition point with the wall temperature leads to a seemingly abrupt jump in local heat flux seen by a fixed surface point. As a result, the $\dot{q} - T_w$ curve for the surface point has almost a local discontinuity (i.e. locally non-differentiable). This causes HTC to overshoot around the transition point, as indicated in Fig. 4.23. This characteristic of HTC around the transition point has been observed for different temperature ratios and temperature differences tested. This oscillatory behavior in HTC calculated using the two temperature method can possibly lead to convergence issues when a loosely coupled CFD-FEA iterative method is applied. Also it is now questionable the same nature of the heat transfer coefficient as a descriptor of the local wall heat flux in the transition region.

The huge overshoot of HTC around the transition point underlines the basic issue/uncertainty regarding its definition and working procedure. It can be argued that using a conventional scaling of the heat flux by means of a constant reference temperature would not bring to a spike in the HTC, but at the same time such non-dimensional heat flux cannot be considered representative of the local HTC as discussed in Sec. 3.1. It must be noted as well that the use of T_{aw} to obtain HTC

would result as well in an overshoot in h due to the different location of the transition point at adiabatic conditions.

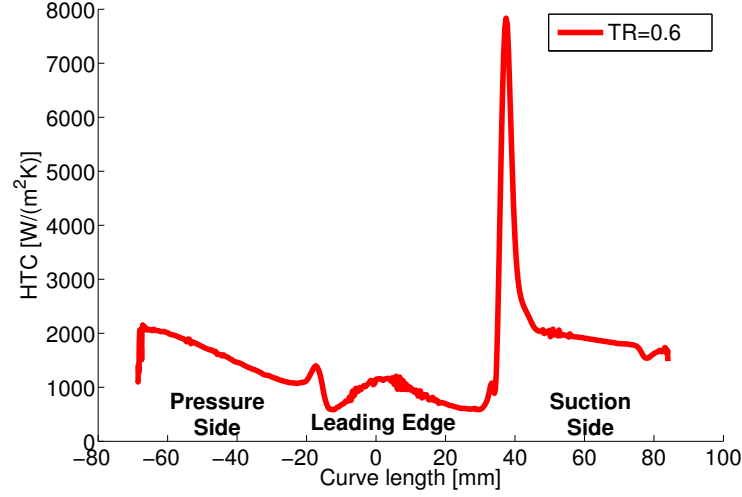


Figure 4.23: HTC plots for TR=0.6 for $k - kl - \omega$ transition model, obtained using the traditional two-point method of Eqn. (3.2).

4.5 Final remarks

The results shown in this chapter and the subsequent discussion lead to the following main observations:

- the aerodynamic and thermal fields in a typical NGV flow are strongly coupled
- due to the dependence of HTC with T_w a classical linear scaling of heat flux with wall temperature may not be adequate for NGV flows
- given the link between aerodynamics and heat transfer, correction of the HTC with TR must account for local flow conditions
- the introduction of a non-fixed transition does exacerbate the limitations of a linear scaling for heat transfer

- the conventional two-temperature approach can result in very oscillatory values of HTC due to transition point dependence on T_w . This would affect not only the consistence and accuracy of HTC itself, but also the stability and convergence of CFD-FEA iterations when HTC is used as the buffer parameter.

The question is now to assess the errors that can derivate from assuming HTC as being aero-determined (as commonly done) in a typical NGV flow as well as to provide a mean to effectively account for the local dependence of HTC with wall temperature. These two issues are discussed in the following chapter.

Chapter 5

An alternative working method for HTC

The results of Ch. 4 highlight the interconnection existing between aerodynamics and heat transfer in compressible NGV flows as well as showing the dependency of HTC with T_w . As discussed in Ch. 2, the topic of HTC dependence with wall temperature has been addressed by some authors, though it seems to be disregarded in many applications.

The purpose of this chapter is to discuss the limitations of available correlations in accounting for the HTC dependence on wall temperature and local flow history.

A newly proposed *three-point method* for heat flux scaling, accounting for HTC dependence on T_w , will be illustrated and its performance, compared with the conventional linear assumption, discussed.

The advantages of using the three-point method under transitional flows will be also presented.

5.1 Comparison with existing correlation

The effect of wall temperature on HTC is normally taken into account when scaling experimental data obtained at high temperature ratios to engine realistically low temperature ratio conditions. As reviewed in Ch. 2, several correlations are available in the open literature. A widely used form for the correction for the Nusselt number is of the kind:

$$Nu = Nu_0(TR)^n \quad (5.1)$$

where Nu_0 is the Nusselt number obtained with $TR = 1$.

One of the most used correlations, developed from numerical and experimental studies on a fully developed turbulent boundary layer on a flat plate, is the one proposed by Fitt et al. (1986). The authors suggest a value for the exponent $n = -0.25$.

Different correlations propose different values for the exponent n (see Ch. 2), depending on the particular flow configuration they are tuned for. This indeed highlights, once more, how the issue of the dependence of HTC non-linearity can hardly be generalized and is dependent on the particular case considered.

A correction of the kind of Eqn. (5.1), regardless of the value of the exponent n , suffers from a fundamental issue: being of global nature cannot account for local flow history. To highlight the shortcomings of this approach when trying to correct Nu with TR on a typical NGV flow, the correlation proposed by Fitt et al. (1986) has been applied to the 2D midspan NGV of Sec. 3.4.1.

A baseline calculation has been made for $TR = 1$. Based on this the Nu distribution for $TR = 0.6$ has been predicted using Eqn. (5.1) with $n = -0.25$. Comparison between Nu distribution obtained by the use of the correlation and the one obtained by CFD are shown in Fig. 5.1. The simulations have been performed in a fully turbulent mode for both the suction and pressure surfaces as the correlation is valid for turbulent boundary layers.

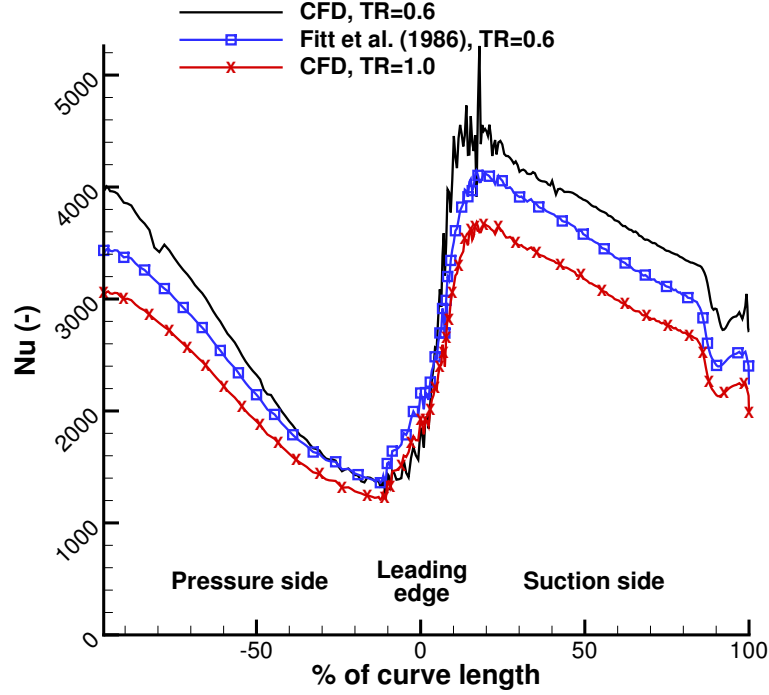


Figure 5.1: CFD results for $TR = 0.6$ case compared with Nu obtained using correlation bt Fitt et al. Fitt et al. (1986).

The comparison of Fig. 5.1 shows the inadequacy of the correlation to correctly scale Nu with TR for the analyzed case. At the leading edge the correlation over-corrects data, while towards the trailing edge a significant under correction is observed. The key reason for the observed discrepancy is that the correlation corrects HTC in a global manner. From the characteristics identified in the present analyses as discussed in Ch. 4, it is clear that the upstream history effect is important. The local nature of the history effect means that a global correction based approach would be inherently inadequate. It should however be recognized that these existing correlations have been useful in giving qualitatively right trend, to a large extent helping to get closer to the HTC at a realistic TR conditions. These correlations also serve as a useful platform and reference point to identify, understand and develop better prediction methods for the problem.

Given the fact that available correlations fail to correctly account for the local dependency of HTC on flow history, the question is to try to obtain a better tool,

based on a *local* correction of HTC with wall temperature. This issue will be discussed in the following section, proposing a novel non-linear *three-point method* for heat flux scaling.

5.2 The three-point method for heat flux scaling

Recalling what said in Sec. 3.1, in the case of a non-linear heat flux curve, the h values obtained using of a two-point method cannot be considered anymore valid over a range of wall temperatures. The idea behind the newly proposed method for HTC correction has been to account for the dependence of h on T_w , allowing to predict heat transfer levels over a range of T_w with a minimum extra computational effort with respect to the previously used two-point method.

In general, both h and T_{aw} are variable with the wall temperature rather than constant, implying the Newton's law of cooling in the form of:

$$\dot{q} = h(T_w)[T_w - T_{aw}(T_w)] \quad (5.2)$$

In the proposed method a linear dependency of h with T_w has been hypothesized. Also since, as already discussed in Sec. 4.2.2, the adiabatic wall temperature showed markedly smaller variations with the wall temperature than the HTC (see Figs. 4.1, 4.8), its dependency on T_w has been neglected ¹. The proposed method considers a modified form for Newton's law of cooling:

$$\dot{q} = \left(h_0 + h_1 \frac{T_w}{T_\infty} \right) (T_w - T_{aw}) = (h_0 + h_1 TR)(T_w - T_{aw}) \quad (5.3)$$

¹In a more general case the proposed method lumps the variability of T_{aw} on h . Its accuracy to predict HTC at different wall temperatures has been also tested recently on a case where high variability of T_{aw} is expected Zhang & He (2014). The model showed comparable accuracy (and a more robust computational stability) with respect to a higher order non-linear method including wall temperature dependency also for T_{aw} .

This is a non-linear equation where h_0 , h_1 and T_{aw} are the unknowns. Such unknowns are function of the spatial position for a given case. Expanding the formula and defining:

$$C_1 = -h_0 T_{aw} \quad (5.4)$$

$$C_2 = h_0 - h_1 \frac{T_{aw}}{T_\infty} \quad (5.5)$$

$$C_3 = \frac{h_1}{T_\infty} \quad (5.6)$$

for given $\dot{q}$ and T_w , Eqn. (5.3) leads to a linear equation with the three unknowns C_1 , C_2 and C_3 :

$$\dot{q} = C_1 + T_w C_2 + T_w^2 C_3 \quad (5.7)$$

The values of the three unknowns C_1 , C_2 and C_3 can be found by solving a linear system using available solutions of $\dot{q}$ at three wall temperatures. This newly proposed and developed method is thus called as *3-point nonlinear method*. Once the system is solved the heat flux $\dot{q}$ distribution for any unknown T_w can be determined based on Eqn. (5.3).

It is important to highlight that C_1 , C_2 and C_3 are pointwise constants on the blade surface: the correction is therefore local. This local correction capability is a key distinction with respect to available correlations.

It needs also to be emphasized that, in terms of computational cost, only three CFD solutions are required to construct the proposed model. As seen previously, normal practice to extract the HTC distribution on a blade surface requires two CFD calculations at two different wall temperatures. This approach is however rigorously valid only if a linear behavior of Newton's equation is observed. In the case of a non-linear behavior the validity of a two-point based approach shrinks to a surrounding of the analyzed T_w . The proposed three-point method, that can be seen as a non-

linear curve fitting of the heat flux curve, is valid instead for a whole range of wall temperatures.

The method does not require any numerical approximation techniques. It is purely analytical, simple with very little extra processing cost. The following sections show applications of the three-point method to the scaling of wall heat flux with TR for the 2D midspan section as well as the 3D MT1 NGV. It will then be considered how the method may help in more challenging cases considering the influence of wall temperature on the laminar-turbulent flow transition.

5.3 Application to 2D blade section

Three data sets corresponding to temperature ratios of 0.95, 0.75 and 0.6 have been used to calculate the constants of Eqn. (5.7). The three-points method has been used to predict $\dot{q}$ for temperature ratios in the range 0.7 to 0.9. The comparison between model results and the direct CFD simulations can be seen in Fig. 5.2. The direct CFD simulations of Fig. 5.2 have been obtained with constant wall temperature corresponding to the desired TR. The proposed model ensures a really good matching of calculated data with the direct CFD results on the whole range of T_w examined.

To highlight the large errors resulting from the assumption of HTC being independent of T_w , the conventional two-points method has been also used to predict heat transfer levels at different temperature ratios. Heat flux distributions corresponding to TR 0.95 and 0.9 have been used to calculate h and T_{aw} . The obtained HTC and adiabatic wall temperature have then been used to calculate, using Eqn. 3.1 with $T_{ref} = T_{aw}$, the heat transfer distributions at temperature ratios 0.8, 0.75 and 0.6. The results are shown in Fig. 5.3. The effect of the increase of h with the TR, shown in Sec. 4.2, is not predicted by the two-points method, resulting in errors in heat flux predictions. Such errors become larger moving away from the original wall tempera-

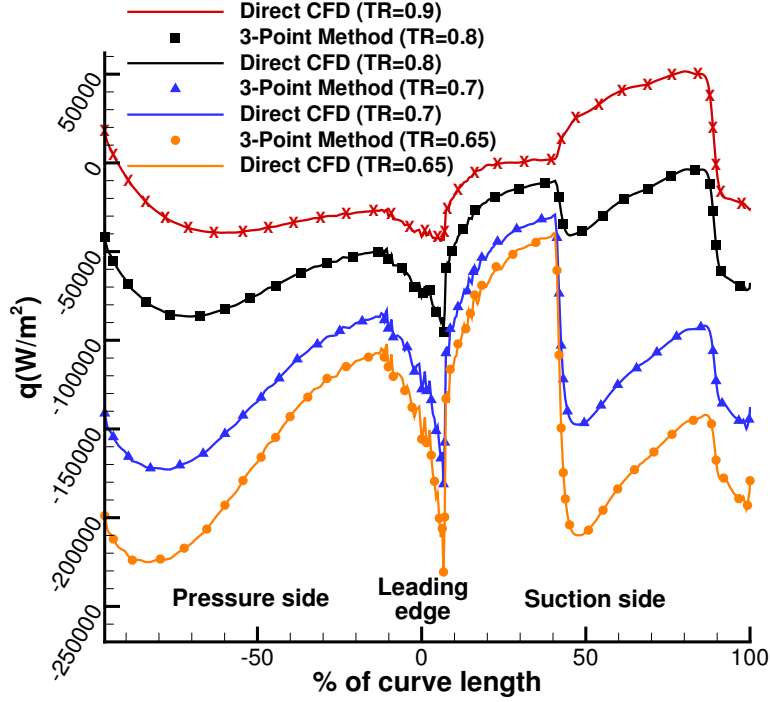


Figure 5.2: Heat flux comparisons between the direct CFD solutions and those constructed from the 3-points model.

tures considered. Figure 5.3 is a clear example of how the validity of the two points method shrinks if the dependence of HTC on T_W becomes non-negligible.

5.4 Application to full 3D heat flux

For the 3D case, the three-point method has been applied to predict data at TR=0.6. The constants of Eqn. 5.7 are calculated using CFD solutions at TR 0.99, 0.8 and 0.7. In the same fashion the two-points linear method is applied to extrapolate the heat flux distribution on the blade and endwalls at TR=0.6, using h and T_{aw} calculated at the quasi-adiabatic case (TR=0.99). Figure 5.4 shows the comparison between the direct CFD results and the two used prediction methods (the 2-point linear and the 3-point non-linear). The improved predictive performance of the three-point method with respect to the conventional two-points one is evident, especially in the regions characterized by highly 3D flows (corresponding, as seen previously, to high HTC

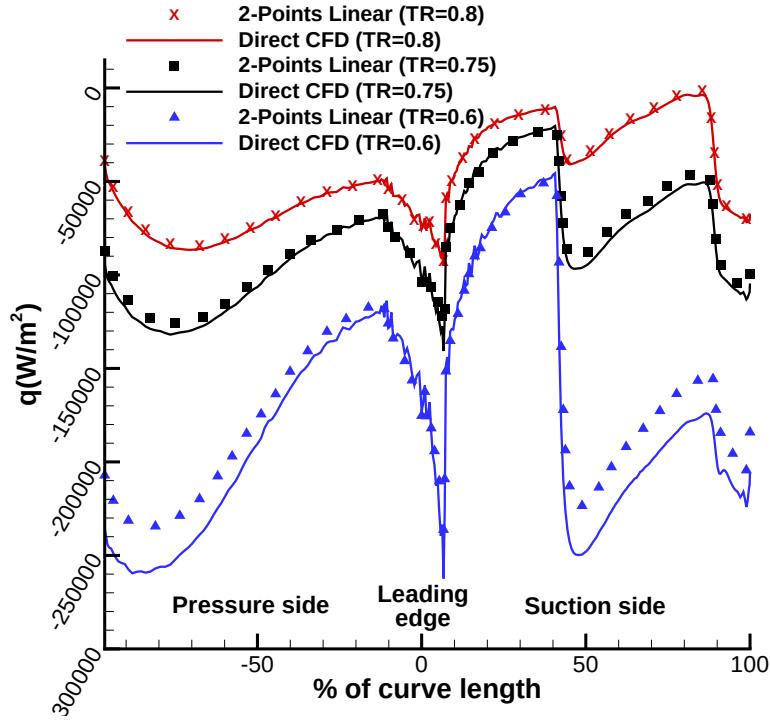


Figure 5.3: Heat flux comparisons between the direct CFD solutions and those constructed from the linear 2-points model.

non-linearity). To better compare the predictive performance of the two models, the percentage errors of the predictions are calculated using Eqn. (5.8).

$$\%Error = \left| \frac{\dot{q}_{Predicted} - \dot{q}_{CFD}}{\dot{q}_{CFD}} \right| * 100 \quad (5.8)$$

Comparison of the errors is shown in Fig. 5.5. Heat transfer errors obtained using the conventional two-point method can be as high as 25% in some regions. Using the proposed three-point method allows to reduce the prediction errors significantly. These comparisons demonstrate consistently that the new three-point method offers, at the price of a 50% increase in computational time compared to the 2-point method, the possibility of much more accurately predicting the heat flux over a whole range of wall temperatures.

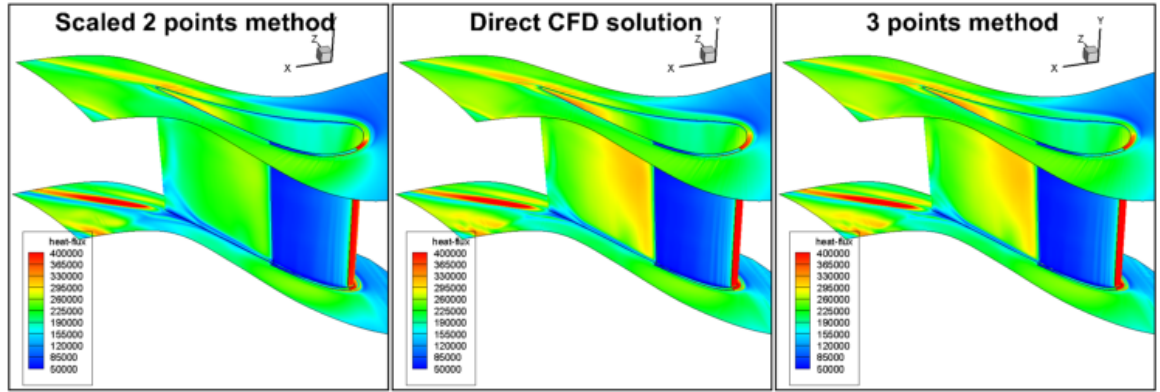


Figure 5.4: Wall heat transfer (W/m^2) for $TR = 0.6$, direct CFD (center), conventional 2-point method (left) and new 3-point method (right).

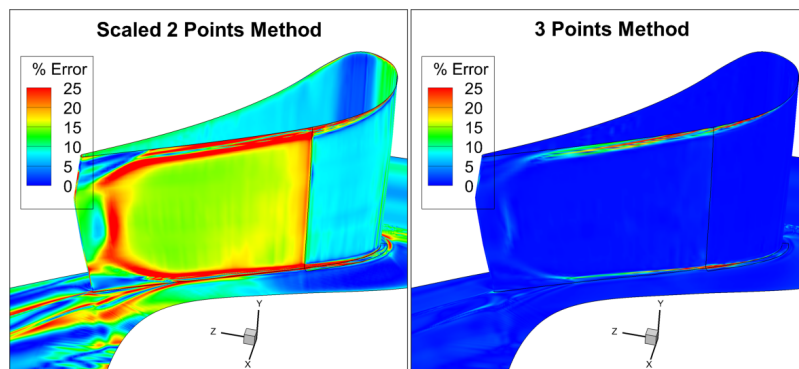


Figure 5.5: Prediction error comparison between the two-point and three-point.

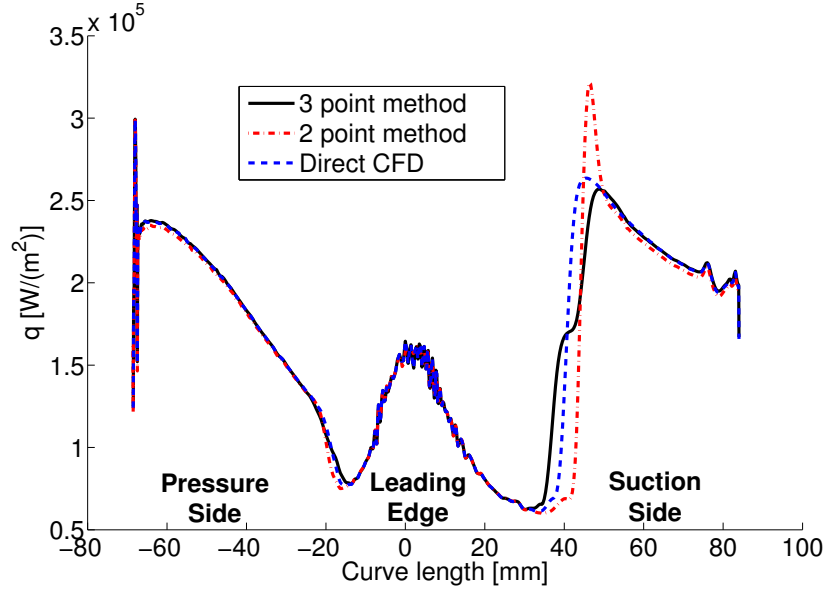


Figure 5.6: Comparison of the new 3-point method and the conventional 2-point method for a transitional case using the $k - kl - \omega$ transition model.

5.5 Application to transitional flows

Given the previously highlighted limitations of the conventional two-point method when the transition point is not fixed and the implications in accuracy and stability when iterative CFD-FEA methods are of interest (see Sec. 4.4), the 3-point method has been applied on such case as well.

The two-point method has been used to linearly extrapolate heat transfer data obtained on $TR=0.8$. The three-point method instead has been used based on CFD data at $TR=0.99$, 0.8 and 0.6 . Predictions by both models are then made at $TR=0.7$. The results compared to that of the direct CFD at $TR=0.7$ are shown in Fig. 5.6.

It is clear that the conventional two-point method results in a very erroneous prediction of the transition point (predicted to be in the same place as for $TR=0.8$) and a large overshooting of the heat flux around the transition. The overshooting is a result of the local non-differentiability of the heat flux around the transition point, generating a spike in the local HTC. However, the new three-point method is able to follow much more closely the direct CFD solution. The advantage of the 3-point

method in terms of enhanced robustness can clearly be seen here, as no overshooting in the heat flux curve can be observed as opposed to the two-point method. The enhanced robustness of the three-point method is chiefly attributed to its multi-point curve-fitting for a range of T_w . On the other hand, the conventional two-point method is based on a finite difference approximation of a local differential at a single T_w point, which is expected to experience difficulties around the transition with a local non-differential behavior.

5.6 Concluding notes on the use of both two and three-point methods

The proposed three point method represents a fundamental departure from the traditional two-point method for HTC calculation. The author believes it is fundamental to stress out, once again, that the key difference between the two methods lays in the fact that the three-point method is, fundamentally, a curve fitting tool, while the second one is based on a finite differencing approach, local by nature.

As stressed out in the previous chapters, it is clear that the use of the two point method can be considered representative of the local HTC if and only if it is used in a finite difference fashion. This is a key observation stemming out of the results of the first part of this thesis and it is a conclusion that the author would like to stress out once again as it constitutes a fundamental message of the thesis.

It is common practice to use the traditional two point method on a much higher ΔT than a finite differencing operation would require. This approach does result in errors if non-linear effects are to be expected. However, due to the local dependency of HTC on previous flow history, the magnitude of this error is case dependent, and can only be estimated a posteriori. To this authors belief, general guidelines for a reliable use of the two point method in its common used form (where ΔT is chosen

arbitrarily and not close to zero) would be hard to establish due to the specificity of the problem. This is particularly critical considering that the tolerances on admissible errors on metal temperatures are extremely low if reliability is of concern.

The same considerations apply when trying to establishing the maximum amount of linear extrapolation acceptable when linear scaling of the heat flux is performed using HTC calculated with $\Delta T \rightarrow 0$.

The three-point method, being a curve fitting tool, is naturally valid on a range of wall temperatures, allowing to avoid all the above mentioned shortcomings of the traditional two points method. Conversely to the two-point method, where a small ΔT should be used to establish a local HTC, the author recommends the use of the three-point method by selecting the three temperatures needed to calculate the unknowns in Eqn. (5.3) in order to span the whole range of wall temperatures of interest. Care should be taken not to select the three temperatures so close that non-linear effects won't be observable as the three point method will then degenerate in a linear fitting.

5.7 Final remarks

In this chapter a novel method to locally correct HTC with T_w has been developed and its predictive capabilities tested on the analyzed NGV.

The fundamental characteristics of such method are:

- unlike the traditionally used correlations that perform a global scaling of HTC with TR, the proposed method applies a local correction accounting for local flow history
- the method has a broader range of application than correlations, strictly valid only for the particular configuration they have been tuned to

- unlike the traditional two-point method, allowing to calculate HTC for a single T_w when non-linearity of the HTC is observed, the three-point method allows to scale the wall heat flux for a whole range of T_w
- the method can improve the stability of iterative CFD-FEA algorithms when applied to transitional flows with a non-fixed transition point

The proposed three-point method can be an effective mean to account for the local dependence of HTC with wall temperature in a more consistent way than what currently done. The method has been recently adopted by other authors to account for HTC dependence with T_w (see Zhang & He (2014), Liu et al. (2015), Liu et al. (2014), Lavagnoli et al. (2014), Caloni & Shahpar (2015)) and has been integrated in Rolls-Royce thermal analysis methods.

This said, the proposed three-point method should only be seen as an improvement of the current HTC scaling tools, rather than an accurate modeling tool. A full appreciation of the effects of the link between aerodynamics and heat transfer can only be obtained through CHT calculations. This is particularly important in the development of modern turbines, where a 5 K difference in metal temperature (corresponding to less than 1% error in a typical NGV) can effectively half/double the life of the component.

Given the importance of a correct prediction of metal temperatures in modern gas turbines and the inherent limitations of simple corrections when high accuracy is sought, in the following part of this thesis the development, validation and application of a CHT solver will be discussed.

Part III

Conjugate Heat Transfer development

Chapter 6

Baseline RANS solver

The connection existing between aerodynamics and heat transfer, as shown in the first part of this thesis, highlights the need to resort to design and analysis tools capable of accounting for this interaction. For this reason the modeling capabilities of Rolls Royce Hydra CFD solver have been extended with the implementation of a steady CHT solver.

The original Hydra solver has been modified to allow for the coupled solution of the fluid and solid domains.

This chapter serves as an introduction to the working principles of Hydra flow solver, describing Hydra's numerical features that are directly connected to the development of the CHT solver, while the details concerning the implementation of the CHT solver will be discussed in Ch. 7. Attention will be limited to Hydra steady solver as unsteadiness has not been considered throughout the work.

A full description of Hydra numerical schemes is outside the scope of this thesis and can be found in Moinier (1999).

6.1 Governing equations

Three dimensional unsteady Navier Stokes equations can be represented in compact form as:

$$\frac{\partial Q}{\partial t} + \nabla \cdot \mathbf{F} = 0 \quad (6.1)$$

where the vector Q contains the conservative flow variables while $\mathbf{F}$ describes the convective and diffusive terms of the equations. No volume sources are considered in the present dissertation.

$$Q = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho e \end{pmatrix} \quad (6.2)$$

$$F_x = \begin{pmatrix} \rho u \\ \rho u^2 + p - \tau_{xx} \\ \rho uv - \tau_{yx} \\ \rho uw - \tau_{zx} \\ (\rho e + p)u - u\tau_{xx} - v\tau_{yx} - w\tau_{zx} - kT_x \end{pmatrix} \quad (6.3)$$

$$F_x = \begin{pmatrix} \rho v \\ \rho uv + p - \tau_{xy} \\ \rho v^2 + p - \tau_{yy} \\ \rho vw - \tau_{zy} \\ (\rho e + p)v - u\tau_{xy} - v\tau_{yy} - w\tau_{zy} - kT_y \end{pmatrix} \quad (6.4)$$

$$F_x = \begin{pmatrix} \rho w \\ \rho uw - \tau_{xz} \\ \rho vw - \tau_{yz} \\ \rho w^2 - \tau_{zz} \\ (\rho e + p)w - u\tau_{xz} - v\tau_{yz} - w\tau_{zz} - kT_z \end{pmatrix} \quad (6.5)$$

Equation 6.1, as applied to compressible laminar flows, represents a system of five equations and six unknowns. Closure of the problem is made possible using the perfect gas equation of state:

$$p = \rho RT \quad (6.6)$$

In Hydra $\gamma = 1.4$, μ is evaluated through Sutherland's law and $Pr = 0.7$.

6.2 Space discretization

To allow for a numerical solution, Eqn. (6.1) must be discretized both in space and time. The spatial discretization is achieved through the use of FV method.

Finite Volume methods are based on the integral form of NS equations, rather

than the differential form of Eqn. (6.1).

Considering a generic control volume (CV) V_j , integration of Eqn. (6.1) over the volume and application of the divergence theorem, allows to write the integral form of Navier Stokes equations as:

$$\frac{\partial}{\partial t} \int_{V_j} Q dV_j + \oint_{\partial V_j} \mathbf{F} \cdot \mathbf{dn} = 0 \quad (6.7)$$

Hydra uses a cell-vertex scheme with *median-dual* (Barth (1994)) control volumes. Flow variables are stored in the grid vertices and control volumes are constructed around each grid vertex, connecting the centroids of the surrounding elements, face-centroids and edge-midpoints (Fig. 6.1). This approach ensures that the mesh control volumes have perfectly matching boundaries and do not overlap.

A control volume V_j is built around each mesh node j , as shown in Fig. 6.1. To construct the space-discretized form of Eqn. 6.7, an average value $\bar{Q}_j$ of the flow properties over the control volume is defined as:

$$\bar{Q}_j \equiv \frac{1}{V_j} \int_{V_j} Q dV_j \quad (6.8)$$

and Eqn. 6.7 can be expressed in its space-discretized form as:

$$\frac{\partial \bar{Q}_j}{\partial t} + \overbrace{\frac{1}{V_j} \sum_i^{R_j} \mathbf{F}_{ij} \cdot \Delta \mathbf{S}_{ij}} = 0 \quad (6.9)$$

Here the surface integral has been turned into a finite summation over the boundary surfaces of the j -th control volume. The second term of the LHS is also referred to as the residual on the j -th volume R_j . The quantity $\mathbf{F}_{ij} \cdot \Delta \mathbf{S}_{ij}$ represents the flux over the surface between nodes i and j , and $\Delta \mathbf{S}_{ij}$ is the vector normal to the CV surface. In the following the term *edge weight* will be also used to refer to $\Delta \mathbf{S}_{ij}$. Included in the summation are the flux contributions of boundary nodes. The treatment of flow

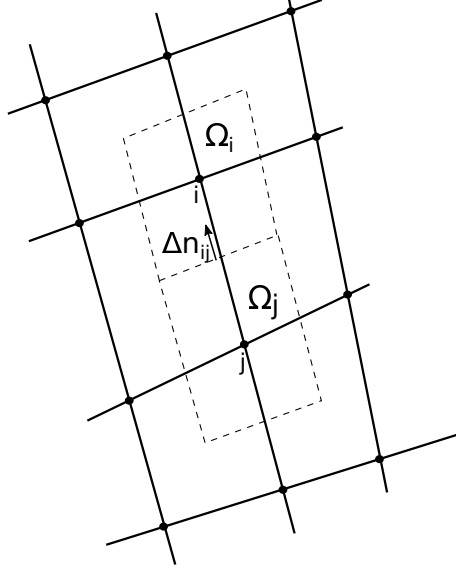


Figure 6.1: Control volumes Ω_i and Ω_j (dashed) built around i-th and j-th mesh nodes.

boundary conditions (BCs) in the original solver will be briefly discussed in Sec. 6.2.3. The edge weights are always pointing to the outward normal to the CV. This ensures that $\Delta \mathbf{S}_{ij} = -\Delta \mathbf{S}_{ji}$ and conservation of fluxes on the whole domain is enforced by construction.

It is convenient to decompose the cell residual R_j in two distinct contributions: a convective contribution and a dissipative¹ one. It is thus possible to write:

$$R_j = \frac{1}{V_j} \sum_i (\mathbf{F}_{ij}^C(Q_{ij}) + \mathbf{F}_{ij}^D(Q_{ij}, \nabla Q_{ij})) \cdot \Delta \mathbf{S}_{ij} \quad (6.10)$$

This distinction is not purely formal: convective terms and dissipative terms represent two different physical phenomena that require different numerical modeling.

As shown in Eqn. 6.10, calculation of the residual in a control volume requires the evaluation of the flux at its boundaries. The value of flow variables is however not available at CV interface, but only at mesh vertices. The issue is how to calculate fluxes at the volume boundaries based on the solution stored in the mesh nodes. The

¹The more general term *dissipative fluxes* has been preferred by the author in spite of the more common term of *viscous fluxes* as, considering the CHT solver, diffusion is not only caused by viscosity.

approach used is different for convective and dissipative fluxes and will be briefly explained in the following sections.

6.2.1 Convective fluxes

To calculate the numerical convective flux $\mathbf{F}_{ij}^C$ at a CV interface Hydra uses a second-order scheme that is based on Roe's flux-differencing method (Roe (1981)). Such method is based on solving an approximate Riemann solver in the control volume to obtain the flux at the interface.

Given the nodal values Q_i and Q_j , and calling Q_{ij}^i and Q_{ij}^j the *reconstructed* solution at either side of the ij interface, the original formulation of Roe's method allows to calculate $\mathbf{F}_{ij}^C$ as:

$$\mathbf{F}_{ij}^C(Q_{ij}) = \frac{1}{2} \left(\mathbf{F}_{ij}^C(Q_{ij}^i) + \mathbf{F}_{ij}^C(Q_{ij}^j) \right) - \frac{1}{2} \left| \frac{\partial \mathbf{F}_{ij}}{\partial Q_{ij}} \right| (Q_{ij}^i - Q_{ij}^j) \quad (6.11)$$

The details of the derivation of Eqn. 6.11 are outside the scopes of this dissertation. It can intuitively be observed, however, that Roe's method calculates the flux as a sum of wave contributions on both sides of the interface. Equation 6.11 represents a centered difference evaluation of the flux with an additional dissipative term that depends on the Jacobian of the fluxes. The last term of Eqn. 6.11, together with adding numerical dissipation to the convective flux, accounts for the flow characteristic direction allowing for an upstream discretization. Due to its dissipative nature, the term $\frac{1}{2} \left| \frac{\partial \mathbf{F}_{ij}}{\partial Q_{ij}} \right| (Q_{ij}^i - Q_{ij}^j)$ is included in the dissipative fluxes $\mathbf{F}_{ij}^D$. The reason for this will be clear when dealing with the time integration (Sec. 6.3).

The reconstruction of the flow variables at the CV boundaries is computationally expensive. In Hydra Eqn. 6.11 is modified so to evaluate the convective fluxes in the first term at the nodes instead of at the interface. In such way evaluation convective fluxes does not require variable reconstruction. The computational saving is again

connected to the used Runge-Kutta scheme for time integration.

Reconstruction of the values of flow variables inside the volume is performed using the Monotonic Upstream-Centered Scheme (MUSCL) approach of Van Leer (1979). A second order interpolation is used to obtain the solution at the CV boundaries.

One of known problems for Roe's scheme in its original formulation is that the dissipation term vanishes as one, or more, of the eigenvalues of the solution are zeroed (see Eqn. 6.11). This can however lead to non physical expansion shocks, allowing for a decrease in flow entropy. To correct for this Hydra ensures that a certain minimum amount of dissipation is always maintained by applying the Harten entropy correction Harten (1983) to the flux eigenvalues.

The numerical dissipation term is further modified in Hydra implementation to allow a high resolution capturing of shocks.

6.2.2 Dissipative fluxes

As diffusion is an isotropic process, the discretization of dissipative terms cannot be carried out with the same upwind approach used for the convective fluxes.

Calculations of the nodal gradient of variables is carried out making use of Green-Gauss theorem.

$$\int_{V_j} \nabla Q_j d\Omega = \oint_{\partial V_j} Q_j dS \quad (6.12)$$

A FV discretization of Eqn. 6.12 allows to calculate ∇Q_j as:

$$\nabla Q_j = \frac{1}{V_j} \sum_i Q_{ij} \mathbf{s}_{ij} \quad (6.13)$$

where the values of the flow variables at the interface of the CV are obtained as average of the quantities either side of the face:

$$Q_{ij} = \frac{1}{2}(Q_i + Q_j) \quad (6.14)$$

Evaluation of the dissipative fluxes requires also to know the values of gradients at the CV faces. Again this is done through an arithmetic average:

$$\overline{\nabla Q}_{ij} = \frac{1}{2}(\nabla Q_i + \nabla Q_j) \quad (6.15)$$

This brings to a central differences stencil for the discretization of diffusive fluxes. Such scheme is second order accurate on smooth grids.

Application of Eqn. (6.15) is very appealing from the computational point of view as only values of the variables at grid nodes are used to evaluate gradients on the volume faces. This approach has however a problem due to the resulting stencil of the Laplace operator bringing to poor damping performance of high frequency modes (Haselbacher & Blazek (2000)).

Assuming a 1D grid of evenly-spaced points, using a notation typical of structured grids where points i and $i + 1$ are consecutive points, it is easy to demonstrate how the use of Eqn. (6.15) brings to a discrete Laplacian in the point i in the form:

$$\nabla^2 Q_i = f(Q_{i-2}, Q_i, Q_{i+2}) \quad (6.16)$$

Due to this odd-even decoupling the numerical scheme will fail to damp high frequency error modes. This can be easily seen by considering a sawtooth solution as the one represented in Fig. 6.2. In this case a discretization of the Laplacian of the kind of Eqn. (6.16) will 'see' a constant solution, resulting in a null Laplacian. This issue becomes particularly critical in the solid domain calculations where the numerical dissipation terms of the convective fluxes are absent. A typical modification to Eqn. (6.15) ((Blazek 2005) consists of replacing the component of the gradient in the direction of the face normal $\mathbf{n}_{ij}$ with a finite difference approximation of the

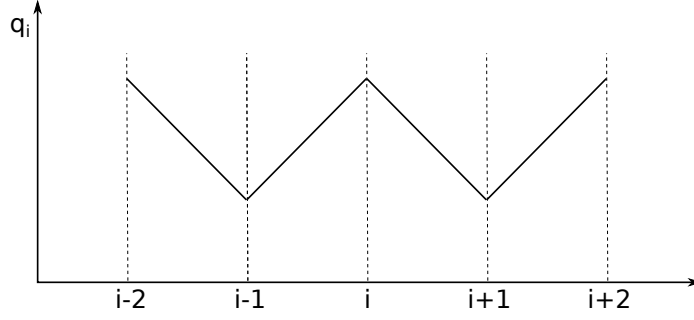


Figure 6.2: Typical sawtooth mode.

derivative across the edge:

$$\nabla Q_{ij} = \overline{\nabla Q}_{ij} - \left[\overline{\nabla Q}_{ij} \cdot \mathbf{n}_{ij} - \frac{\Delta Q_{ij}}{L_{ij}} \right] \mathbf{n}_{ij} \quad (6.17)$$

This formulation gives a more stable stencil for $\nabla^2 Q_i$, including the nodes $i - 1$ and $i + 1$. This allows a better damping performance of high frequency modes. The reason for this better damping when including nodes $i - 1$ and $i + 1$ in the Laplacian stencil can be easily seen if considering again the sawtooth mode of Fig. 6.2. Such discretization is used in Hydra and has been retained also for discretization of dissipative fluxes in the solid solver.

6.2.3 Boundary conditions

Before marching the solution in time, flux contribution of the boundary conditions must be added to the residual in the boundary nodes. In the following the boundary conditions used for the simulations performed during this thesis work will be briefly discussed. A separate discussion in Sec. 7.3 of the following chapter will be dedicated to the treatment of solid-fluid thermal boundary interface in the CHT solver.

- **Viscous wall** On viscous walls the velocity components are explicitly set to zero. If the wall is isothermal the temperature is imposed on wall nodes. The corresponding residual of momentum equation is zeroed, as well as the residual

on the energy equation in case of constant temperature wall. For adiabatic walls no temperature is specified and no boundary flux is added to the energy equation.

- **Subsonic pressure inlet** Values of total pressure, total temperature, two flow angles and turbulent quantities are specified at subsonic pressure inlets. To calculate the boundary flux corresponding to these freestream conditions the concept of a *virtual node* (Fig. 6.3). Virtual nodes are purely abstract, meaning that they are not physically existing in the mesh. The boundary conditions values are assigned to the virtual node and, based on these values, the flow variables in the virtual node are computed by extrapolation from the flow properties at the boundary node. The boundary flux is then computed as in Eqn. (6.11) between the dummy node and the boundary node.
- **Pressure outlet** Boundary flux is calculated similarly than in the case of the subsonic pressure inlet. For pressure outlet only the static pressure is specified. The other flow quantities are again calculated by extrapolation from the fluid domain.
- **Periodic** Periodic boundaries in Hydra must be conformal: nodes on both sides of the periodic boundaries are exactly matched. Indicating with p_1 and p_2 two corresponding periodic nodes on the two periodic boundaries, the residual is ensured to be identical for both nodes and equal to the sum of the residuals in the single nodes. This in practice corresponds to considering the two nodes as a single node, belonging to a bigger control volume and ensures by definition the periodicity of the solution.

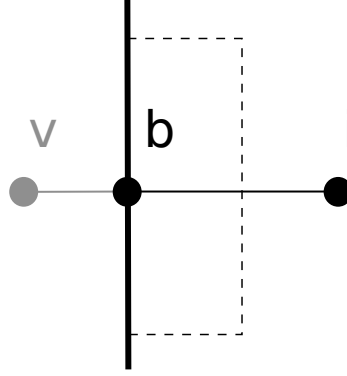


Figure 6.3: Virtual node v, boundary node b, internal node i.

6.3 Time discretization

Time integration of Eqn. (6.7) is carried out with an explicit time marching method using the five step Runge-Kutta smoother of Martinelli (1987).

The solution at the time step Q_j^n is marched in time according to the following:

$$\begin{aligned} Q_j^{(0)} &= Q_j^n \\ Q_j^{(k)} &= Q_j^n - \alpha_k \Delta t_j R_j^{(k-1)}, \quad k = 1, \dots, 5 \\ Q_j^{n+1} &= Q_j^{(5)} \end{aligned} \tag{6.18}$$

where the residual $R_j^{(k-1)}$ is:

$$\begin{aligned} R_j^{(k-1)} &= C_j(Q_j^{(k-1)}) - B_j^{(k-1)} \\ B_j^{(k-1)} &= \beta_k D_j(Q_j^{(k-1)}) + (1 - \beta_k) B_j^{(k-2)} \end{aligned}$$

The term $C_j(Q_j^{(k-1)})$ represents the advective residual on cell j and $D_j(Q_j^{(k-1)})$ accounts for the dissipative terms, including numerical dissipation for the upwind discretization of convective fluxes. The used values for α_k and β_k are shown in Tab. 6.1.

The advantage of this time integration method is due to the fact that dissipative fluxes are evaluated only in two of the five steps of the method, allowing for a considerable amount of saving in terms of computational time. The time-step Δt_j is

k	α_k	β_k
1	$1/4$	1
2	$1/6$	0
3	$3/8$	$14/25$
4	$1/2$	0
5	1	$11/25$

Table 6.1: Constants used for the 5-step Runge-Kutta time-marching.

calculated in each cell according to the local CFL condition based on local viscous and dissipative characteristic wave speeds.

The time-stepping method described previously shows poor convergence when used on viscous computations. The mismatch between convective speeds and sound propagation speeds in the boundary layer cells increases the stiffness of the governing equations, affecting the convergence rate.

Acceleration of the convergence rate and increase of the stability is achieved in Hydra by modifying the local time-stepping using a block Jacobi preconditioner. Time derivative term in Eqn. (6.1) is pre-multiplied by a preconditioning matrix such that:

$$P^{(-1)} \frac{\partial Q}{\partial t} + \nabla \cdot \mathbf{F} = 0 \quad (6.19)$$

The use of a preconditioner allows to reduce the unbalance between the eigenvalues of the system in boundary layer flows, reducing the stiffness of the problem and increasing convergence rate.

Further details on the preconditioner are omitted here as of limited interest in the context of the CHT solver implementation. The preconditioner used in Hydra flow solver is not used in the developed solid solver.

6.4 Multigrid

Hydra makes also use of a Full Approximation Scheme (FAS) (Brandt (1980)) multi-grid (MG) smoother to increase convergence rate.

Explicit time-marching methods are efficient in damping high frequency errors but are not as efficient with low frequency ones (see Blazek (2005)). The use of multigrid - solving the discretized system over a number of coarser grids - allows to transform low frequency error modes on a fine grid to high frequency modes on a coarse grid that can be efficiently damped by the Runge-Kutta time-marching. Residuals on a fine mesh are transferred to a coarser mesh through a *restriction* step. The inverse process (going from a coarse mesh to a fine one) is named *prolongation*.

The reader is referred to Moinier (1999) or Brandt (1980) for a more detailed description of the MG implementation of the in Hydra. For the purpose of this dissertation no other details are deemed to be necessary.

As will be shown in the following chapter, the solid conduction equation is equivalent to the fluid energy equation with zero velocity, thus the multigrid algorithm used in Hydra has been retained also for the solid solver.

6.5 Overview of Hydra workflow

After introducing all the relevant features of Hydra RANS solver, the internal workflow of the solver, showed in Fig. 6.4, will be discussed in this section. One solver iteration is composed of a full multigrid cycle. At each grid level the Jacobian preconditioner is calculated before the solution is marched in time using the Runge-Kutta smoother.

The time-marching cycle starts with the calculation of the gradients and of dissipative fluxes. These two steps, as mentioned in Sec. 6.3, are only executed at steps 1, 3 and 5 of the Runge-Kutta cycle. Inviscid fluxes are then added to the residuals,

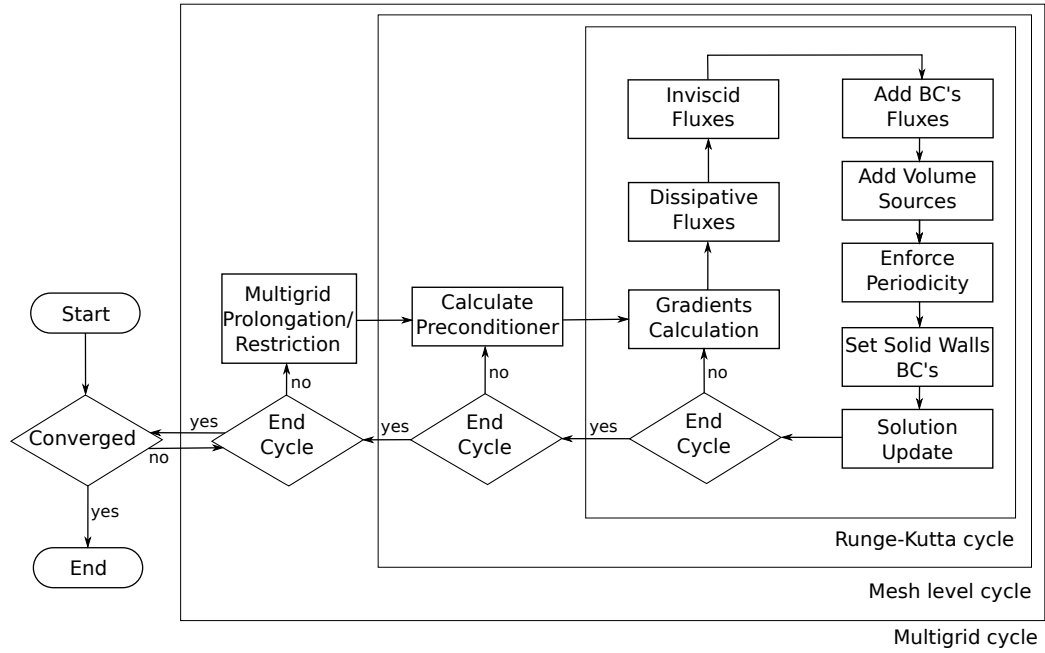


Figure 6.4: General workflow of Hydra solver.

as well as boundary condition fluxes. The addition of volume sources, for the cases of interest in this thesis, only refers to the addition of turbulence volume sources (i.e. production and dissipation terms). The residuals so obtained are then corrected for periodicity and the solid walls boundary conditions are imposed as specified in Sec. 6.2.3. Update of the solution is then performed, based on the nodes residual, through Eqn. (6.18).

The developed conjugate solver retains the same structure of Fig. 6.4, but many of its building functions had to be altered to accommodate for the CHT solution.

Chapter 7

Extension of fluid solver to Conjugate Heat Transfer solver

The simulation capabilities of Hydra have been extended allowing for a *fully coupled* CHT simulation. The solid and fluid domains are merged in a single mesh and the Fourier heat equation is marched in time in the solid domain using the same solver that handles fluid equations.

The details of the implementation and the integration of the solid solver in the original Hydra will be discussed in the present chapter, as well as the additional features implemented to extend the present modeling capabilities of Hydra fluid solver alone.

7.1 Governing equation and its discretization

Ignoring any heat transfer by radiation and volume sources of heat, the evolution of the temperature field in a solid body is described by the heat equation in the form:

$$\frac{\partial T}{\partial t} = \nabla^2(\alpha T) \quad (7.1)$$

where $\alpha = k/\rho C_p$ is called thermal diffusivity of the metal, and k , ρ , C_p are the thermal conductivity, density and thermal capacity respectively.

Considering the set of equations (6.1) to (6.5) it is easy to show that, when all velocity terms u , v and w vanish, the energy equation degenerates into Eqn. (7.1). This allows the solution of the Navier Stokes and heat equations using the same solver.

A FV semi-discretized form of Eqn. (7.1) reads:

$$\frac{\partial \bar{T}_j}{\partial t} + \frac{1}{V_j} \sum_i \nabla(\alpha T)_{ij} \cdot \Delta \mathbf{S}_{ij} = 0 \quad (7.2)$$

The CHT solver uses the same discretization of dissipative fluxes for both fluid and solid domains. Evaluation of the fluxes in Eqn. (7.2) is carried out as described in Sec. 6.2.2. The solid thermal diffusivity α is evaluated locally, allowing to consider temperature-dependent material properties and the presence of multiple solid blocks with different material properties (e.g. Thermal Barrier Coating).

7.2 Solid domain time-marching

As described by He & Oldfield (2011) convective phenomena in the fluid domain and diffusive phenomena in the solid domain have completely different time scales.

Time marching in the fluid domain is restricted by the CFL condition:

$$\frac{\Delta t_f V}{\Delta x_f} < 1$$

Similarly, performing a Fourier stability analysis on Eqn. (7.2), using forward Euler time-marching (see Hirsch (1988)), brings to:

$$\frac{\Delta t_s \alpha}{\Delta x_s^2} < 0.5$$

This allows to calculate a typical ratio between the time-steps in the two domains. In a typical near-wall region, where the convective flow velocity can be represented by the speed of sound, and taking a typical grid spacing ratio $\Delta x_s/\Delta x_f = 1E3$ ¹ it is possible to obtain:

$$\frac{\Delta t_s}{\Delta t_f} \simeq 1E9$$

As a consequence of the time-scale unbalance between the two domains a common time-marching strategy between the solid and the fluid domains is unpractical. The solid time marching uses the same Runge-Kutta method described in Sec. 6.3 but with a local time stepping calculated as:

$$\Delta t_{s_j} = Fo \frac{\Delta x_{s_j}^2}{\alpha_j}$$

Where Fo , the Fourier number, is taken as 0.4 and the cell characteristic length $\Delta x_{s_j}^2$ is taken as the smallest dimension of the cell.

Since the heat equation has no convective terms, the solver does not use any pre-conditioning matrix for the time-marching in the solid.

7.3 Fluid-solid coupling strategy

At the interface between a fluid and a solid domain, when heat transfer is of interest, two fundamental conditions have to be met:

$$T_{w_s} = T_{w_f} \tag{7.3}$$

$$\dot{q}_{w_s} = \dot{q}_{w_f}$$

¹For CHT flows in gas turbines the requirement of $y^+ < 1$ is the reason for the big disproportion between solid and fluid grid wall spacing. This value has been obtained by taking as a reference the values of grid spacing for the CHT case of Sec. 4.2.1.1.

where the subscripts s and f indicate quantities at the solid and fluid side of the interface respectively.

A correct solution of the CHT problem must thus be achieved by enforcing continuity of the temperature at the interface, as well as conservation of energy. Steady CHT solvers can achieve this by means of different strategies (see Verstraete et al. (2007)). Broadly speaking the coupling strategies can be grouped in two categories: *loosely coupled* and *strongly coupled* methods.

Loosely coupled methods use a non-synchronous passing of boundary conditions between the two domains as a coupling strategy. Commonly used coupling strategies differ by the boundary conditions being passed between the two domains. The type of boundary conditions passed and the frequency of communication have an effect on the stability of the iterative procedure, as noted by Verstraete et al. (2007). A possible loosely coupled strategy involves the fluid passing the wall heat flux $\cdot q_w$ to the solid domain and the solid passing back the wall temperature T_w to the fluid.

A typical iterative procedure would read like:

1. Solid T_w is applied to fluid wall
2. Fluid domain is marched in time using T_w
3. Wall heat flux q_w from fluid solution is passed to solid domain
4. Solid domain is solved using wall q_w from fluid solution
5. A new wall temperature T_w is obtained from the solid solution and the procedure is repeated

This kind of approach does not strictly enforce Eqn. 7.3, it has a narrower stability region than a strongly coupled approach as noted by Giles (1997), and it is characterized by a continuous lag between the conditions in the two domains due to a non-synchronous update of the interface condition. A strongly coupled approach,

on the contrary ensures a synchronous exchange of boundary conditions between the solid and fluid domains and explicitly enforces continuity of temperature and heat flux at the interface.

For these reasons a strongly coupled approach has been followed: the solid and fluid domains are marched in time and at each time-step a synchronous update of the thermal interface condition is performed.

The update of the thermal condition of interface nodes is carried out exclusively by enforcing the conservation of heat flux and continuity wall temperature at the interface. To ensure conservation of energy at the interface, the present CHT implementation requires the solid and fluid mesh to be conformal at the interface (i.e. each solid wall node must have a single corresponding node in its same position the fluid side of the CHT wall).

With this assumption, from Eqn. (7.3), evaluating the wall heat flux with a first order finite-difference it is possible to obtain:

$$k_f \frac{T_f - T_w}{\Delta n_f} = k_s \frac{T_w - T_s}{\Delta n_s} \quad (7.4)$$

where T_f and T_s are the temperatures in the near-wall nodes on the fluid and solid domain respectively and Δn_f and Δn_s are the distances of the nodes to the wall (Fig. 7.1). Equation (7.4) is imposed also in the case of a solid-solid interface. This allows to obtain a wall temperature T_w that ensures continuity of temperature and wall heat flux at the interface. The use of a strongly coupled implementation represents a fundamental departure from the loosely coupled CHT implementation in Hydra developed by Lad (see Lad & He (2013)).

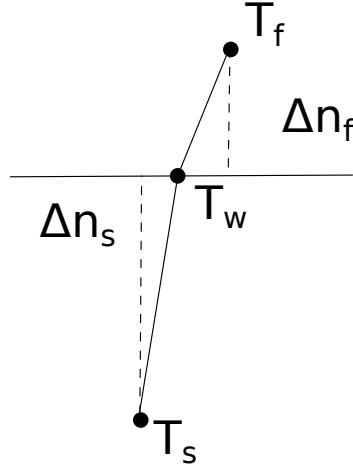


Figure 7.1: Fluid-solid interface nodes.

7.3.1 Coupling strategy for corner nodes

The update approach described so far is however applicable only to wall nodes that are directly connected to a near-wall node through an edge. Such nodes will be referred to as *planar nodes* in the rest of this chapter. Nodes where this condition is not respected will be referred to as *corner nodes*.

An example of corner node, labeled C , is shown in Fig. 7.2, surrounded by planar nodes P_1 , P_2 and P_3 .

While the update in the planar nodes P_i can be carried out using Eqn. (7.8) (e.g. using temperature values in node 1 and 6 for P_1), node C is not directly connected to its near-wall nodes 7 and 4. The 2D case illustrated in Fig. 7.2 is still simple and a strategy can be thought to correctly identify that nodes 7 and 4 are the near-wall nodes corresponding to node C . However, working with an unstructured mesh it is impossible to retrieve such information on more complex 3D corners.² For this reason corner nodes, when present, need to be treated separately with respect to planar nodes. As the information on near-wall nodes is not obtainable, an update of the wall temperature on corner nodes consistent with Eqn. (7.3) is not possible. It has been necessary then to release the condition that wall heat flux should be continuous

²A list of all configurations mapped as corner nodes will be shown in Sec. A.1.

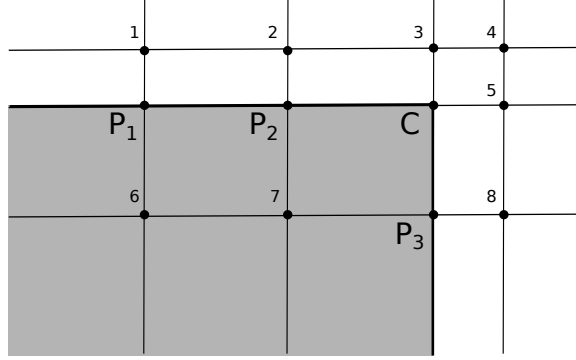


Figure 7.2: An example of corner node situation.

in such nodes. The only condition imposed is the continuity of temperature. The value of T_c in corner nodes is updated based on:

$$T_c = \sum_{i \in E_i} \frac{T_i/d_i}{1/d_i} \quad (7.5)$$

where E_i is the set of wall nodes connected to node C and d_i is the distance between the corner node and node i . For each time step the value of temperature in corner nodes is updated only after all planar nodes have been updated.

It must be stressed out that in many cases the presence of corner nodes can be avoided through a careful blocking of the mesh. Particularly, all validation cases shown in Ch. 8 do not use corner nodes.

7.4 Stability analysis

Stability analysis of the flow solver has been already discussed by Moinier (1999). For this reason only the stability of the solid solver and of the solid-fluid interface condition will be discussed.

7.4.1 Solid solver

Considering a uniform hexahedral mesh, it is easy to show that the discretization of viscous fluxes of Eqn. (6.17) reduces to a classic central second-order discretization of the gradient. It can be shown (see Hirsch (1988)) that such numerical scheme is stable for a Fourier number F :

$$F = \frac{\alpha \Delta t}{\Delta x^2} \leq \frac{B}{4} \quad (7.6)$$

where B is the extension of the stability limit of the time integration scheme over the real axis.

For the used five stage RK scheme $B \approx 9$ (see Pierce & Giles (1997)). Considering however that no influence of the boundary conditions is considered in this analysis and that the space discretization is different for non-uniform grids, a conservative value of 1 for the Fourier number has been used throughout the simulations performed.

7.4.2 Coupling strategy

The stability of the used coupling strategy of Eqn.(7.4) will be discussed in this section, following the approach proposed by Giles (1997).

The analysis is focused on error modes that are propagating in the direction normal to the wall interface. This is a reasonable simplification considering the nature of the wall heat transfer. The analysis is thus carried out considering a 1D domain normal to the wall. As a consequence of this, no convective terms are considered in the fluid domain.

Under these assumptions the numerical system will be stable if:

$$T_{min}^{n+1} \geq T_{min}^n \quad T_{max}^{n+1} \leq T_{max}^n \quad (7.7)$$

where T_{min} and T_{max} are the minimum and maximum temperatures in the whole domain (solid and fluid) and the superscripts n and $n + 1$ refer to two successive timesteps.

If the conditions of stability of the solid and fluid time marching are respected, the relations of Eqn. (7.7) are naturally satisfied in the two domains. It leaves to show that the interface condition of Eqn. (7.4) is compatible with Eqn. 7.7.

Rearranging Eqn. (7.4) it is possible to write the wall temperature T_w as:

$$T_w^{n+1} = \frac{\frac{k_f}{\Delta n_f} T_f^{n+1} + \frac{k_s}{\Delta n_s} T_s^{n+1}}{\frac{k_f}{\Delta n_f} + \frac{k_s}{\Delta n_s}} \quad (7.8)$$

The updated wall temperature results from a weighted average of near-wall fluid and solid temperatures. Since both T_f^{n+1} and T_s^{n+1} satisfy Eqn. (7.7), the wall temperature also will.

It is possible to conclude that the used form of strong coupling is stable, provided that the time marching in the solid and fluid domains are stable as well. The same conclusion can be drawn considering the presence of corner nodes as the temperature update in these nodes is still determined through weighted averaging of nearby nodes.

7.5 Further implemented features

During the developing work on the CHT solver, additional features have been added to Hydra:

- Tripped transition modeling
- Wall heat flux boundary condition
- Convective heat flux boundary condition

The original version of Hydra could only accept Dirichlet thermal boundary condition

at solid walls (specified T_w). The necessity to be able to model realistic turbine cooling cases, where heat flux or convective heat flux boundary conditions are more suitable, brought to the implementation of such capabilities.

The transition model was required because the NGV test case used to validate the CHT solver showed transitional behavior on the suction side. Ensuring a correct matching of the interface heat flux was necessary to successfully conclude the CHT solver validation. To achieve this a tripped transition model was implemented.

The details of the implementation of these additional features will be discussed in the following sections.

7.5.1 Tripped transition modelling

One of the validation tests of the conjugate solver has been performed on NASA C3X nozzle guide vane (Hylton et al. (1983)). Flow around such NGV exhibits transitional behavior on the suction side. Considering the profound influence of transition on the wall heat flux, the inclusion of transition modeling in Hydra has been necessary to successfully conclude the solver validation.

It should be clarified that the objective of the modeling strategy has been to be able to correctly evaluate the heat transfer levels in the transition region for a case where available experimental data, or empirical correlation, allowed to estimate, prior to the CFD run, both the transition point and the length of transition region.

The implemented transition modeling strategy is based thus on a tripped transition approach, where the transition point P_t , extension of laminar region L_{lam} and transition length L_{trans} are specified as inputs by the user rather than being determined as part of the solution. This approach, although not general, revealed to be quite accurate for the purpose and represents still, as pointed out by Denton (2010) a typical approach when dealing with transition.

Turbulence closure in Hydra is achieved through standard eddy viscosity approach.

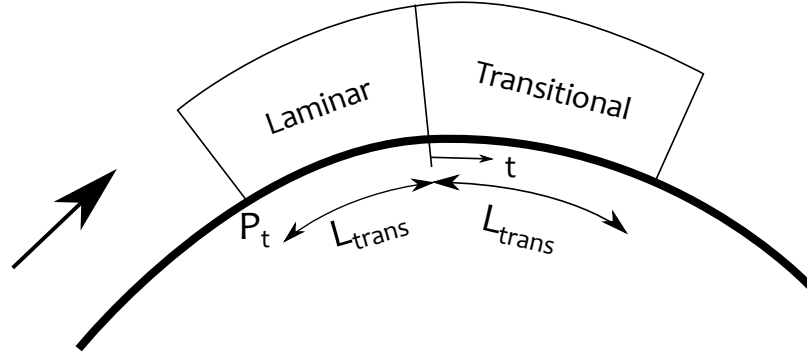


Figure 7.3: Tripped transition modeling.

The turbulence model solves transport equations for turbulence variables, and from these the local values of μ_t are obtained. As illustrated in Fig. 7.3, the used tripped transition approach is based on isolating two regions of the fluid domain: a laminar and a transitional one. The process of identification and mapping of mesh nodes belonging to each of the two zones is performed in the preprocessing stages, based on the user inputs.

Modeling of transition is achieved through the following:

- In the whole computational domain, including the laminar and transitional sub-domains, the turbulence equations are solved as in a fully turbulent case, allowing the calculation of the local values of μ_t
- If in the laminar region, viscous fluxes are calculated assuming $\mu_t = 0$
- In the transitional region, viscous fluxes are calculated using a modified eddy viscosity $\bar{\mu}_t = r(t)\mu_t$, where $0 \leq r(t) \leq 1$

In practice a gradual *ramping* of the eddy viscosity in the transitional region is performed. The ramping coefficient $r(t)$ is function of the local non-dimensional stream-wise coordinate (see Fig. 7.3) and can be seen as an *intermittency-like* parameter, gradually bridging the laminar region to the fully turbulent one. The value of t spans from 0 to 1 from the beginning to the end of the transitional region. The details of the form of the ramping function $r(t)$ are discussed hereafter.

U_{inlet}	5.4	$[m/s]$
ρ	1.2	$[kg/m^3]$
μ	1.8×10^{-5}	$[kg/(ms)]$
μ_t/μ	13.3	$[-]$

Table 7.1: Boundary conditions for the T3A flat plate case.

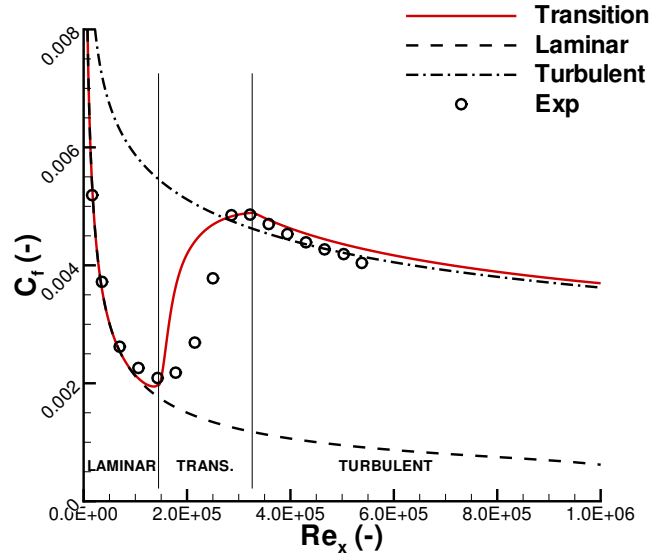


Figure 7.4: T3A test case: tripped transition modeling with linear μ_t ramping.

Calibration of the ramping function $r(t)$ has been performed by comparison with the T3A flat plate test case data obtained by Savill (1996) and Savill (1993)). Boundary conditions for the T3A case are listed in Tab. 7.1. At first a linear ramping function of the kind $r(t) = t$ has been imposed. Figure 7.4 shows the resulting wall skin friction coefficient C_f against experimental data of Savill (1996). The extension of the three specified sub-domains, laminar, transitional and turbulent is also showed.

Figure 7.4 shows that a linear ramping of the turbulent viscosity results in a non-linear increase in C_f . The results were not considered satisfactory and a different function $r(t)$ has been sought with the aim of obtaining a linear-like increase in C_f in the transition region, in line with the observed experiments.

To achieve this, from the simulation of Fig. 7.4 the values of $C_f(t)$ in the transi-

tional region have been extracted and $C_f(0) = C_{f_{lam}}$ and $C_f(1) = C_{f_{turb}}$ obtained. A non-dimensional $\overline{C}_f(t)$ has then been calculated as:

$$\overline{C}_f(t) = \frac{C_f(t) - C_{f_{lam}}}{C_{f_{turb}} - C_{f_{lam}}} \quad (7.9)$$

expressing the local relative increase in friction coefficient. Both $r(t)$ and $\overline{C}_f(t)$ for this linear ramping case are sketched in Figure 7.5(a). The problem has then been to find a suitable ramping function, resulting in a linear $\overline{C}_f(t)$.

This is a typical inverse problem where, based on a desired output (a linear $\overline{C}_f(t)$ increase), the corresponding input ($r(t)$) has to be determined.

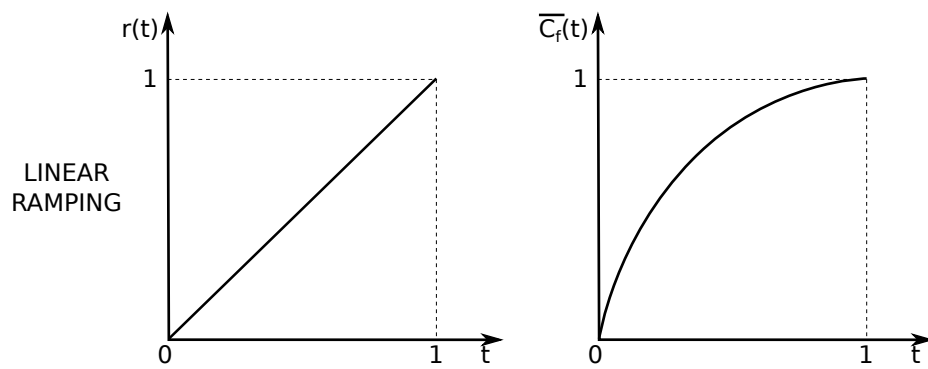
This inverse problem is non-linear, as the relation between $r(t)$ and $\overline{C}_f(t)$ is governed by Navier-Stokes equations. The study of solution methods for such kind of problems is a whole research field in itself and the effort in approaching the problem in a rigorous way was not justified in light of the approximation level of the transition model itself. Here the following practical approach has been followed.

The behavior of the solution in Fig. 7.4 shows that, with a linear ramping $r(t)$, the skin friction grows sharply at the beginning of the transition region and less towards the end. To compensate for this behavior a non-linear $r(t)$ was sought, having a slower ramping in the starting region and increasing towards its end.

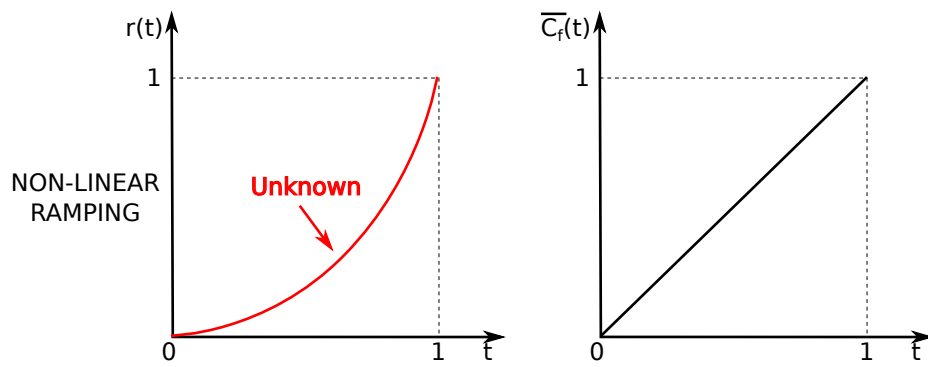
With the help of Fig. 7.5, it can be seen that an ideal candidate for the unknown non-linear ramping function in Fig.7.5 has the shape that corresponds to the obtained $\overline{C}_f(t)$ for the linear ramping, but mirrored with respect to the first quadrant bisector. The curve $\overline{C}_f(t)$ obtained using the linear ramping has then been mirrored, and non-linear curve fitting has been performed to obtain an analytical function approximating the mirrored curve.

$$r(t) = \frac{e^{-41.35t} - e^{4.68t}}{e^{-41.35} - e^{4.68}} \quad (7.10)$$

Equation (7.10) is the final form of the ramping function used. A second test has



(a) Linear ramping



(b) Non-linear ramping

Figure 7.5: Schematics of the linear and non-linear ramping.

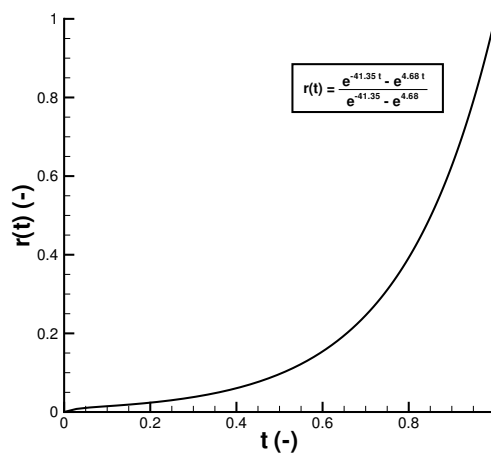


Figure 7.6: Final form of the ramping function used.

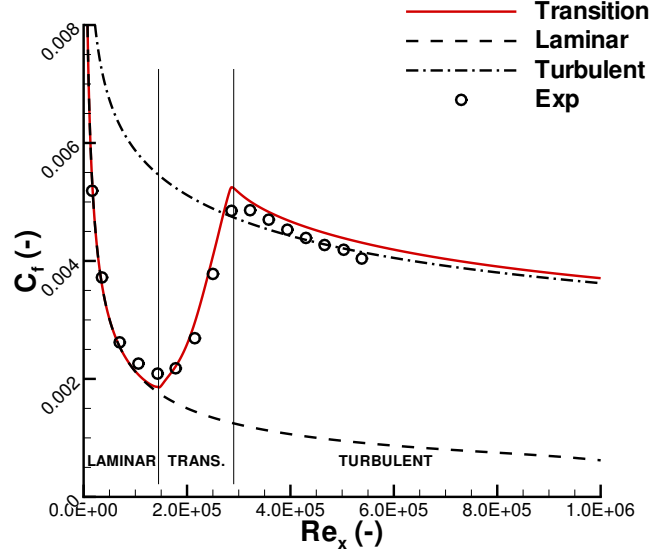


Figure 7.7: T3A test case: tripped transition modeling with non-linear μ_t ramping.

been then performed on the T3A case. Results are shown in Fig. 7.7. With respect to Fig. 7.4 a sharper discontinuity can be observed in the skin friction at the end of the transition region. This is due to the steeper increase in μ_t in the non-linear ramping case before the end of the transitional region. Besides this, a better agreement with experiments was observed and the trend of increase in C_f during transition is captured with satisfactory accuracy.

The used eddy viscosity ramping equation has been also tested on the MT1 NGV. Results of the simulations are shown in Fig. 7.8.

7.5.2 Heat flux and convective thermal boundary conditions

Specifying the wall heat flux is equivalent to imposing the wall temperature gradient (Neumann boundary condition). By using a first order approximation of the gradient operator, the wall heat flux can be expressed as:

$$\dot{q}_w = k \frac{T_w - T_{nw}}{dn} \quad (7.11)$$

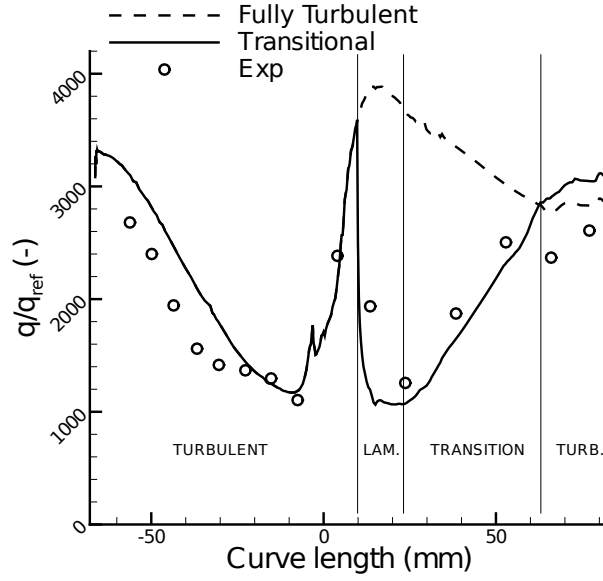


Figure 7.8: MT1 test case: tripped transition modeling with non-linear μ_t ramping.

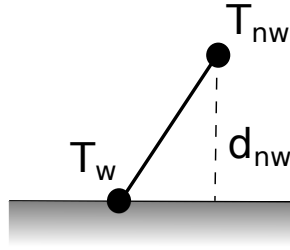


Figure 7.9: Specification of wall temperature gradient through near-wall nodes.

where the subscript nw refers to the near-wall node. This equation can be re-arranged as:

$$T_w = T_{nw} + \frac{\dot{q}_w d_n}{k} \quad (7.12)$$

with symbols as from Fig. 7.9. By using Eqn. (7.12), it was possible impose a wall heat flux by performing an update of the wall temperature, thus requiring minimum modification of the code.

Implementation of the convective boundary condition (also called *third type* or Robin boundary condition) follows the same approach. In a convective thermal boundary condition the heat transfer coefficient h and a driving temperature T_{ref}

are specified. Wall heat flux is then calculated using Newton's law of cooling as:

$$\dot{q}_w = h(T_{ref} - T_w) \quad (7.13)$$

Using Eqn. (7.12) the wall temperature can be expressed as:

$$T_w = \frac{\frac{h}{k} \frac{dn}{dn} T_{ref} + T_{nw}}{\frac{h}{k} \frac{dn}{dn} + 1} \quad (7.14)$$

At each iteration the temperature in the wall nodes is updated using the temperature in the near-wall nodes, according to Eqn. (7.12) for heat flux boundaries or Eqn. (7.14) for convective heat flux boundaries.

The validation tests for the implemented boundary conditions are presented in Ch. 8.

Chapter 8

Validation of the implemented solver

In this chapter the validation tests performed on the implemented solver will be presented. The structure of the chapter reflects the procedure adopted to validate the solver.

At first the aerothermal predictive capabilities of the original Hydra fluid solver have been assessed on a flat plate flow for both laminar and turbulent regimes (Sec. 8.2). This allowed to obtain a reference benchmark to judge the quality of the validation results for the the implemented thermal boundary conditions, discussed in Sec. 8.3.

The accuracy of the implemented solid conduction solver has then been analyzed. Hydra results have been compared against the analytical solution of solid conduction in a hollow cylinder as well as a finite element solution using ANSYS Mechanical on a L-shaped beam. The results of the solid solver validation are discussed in Sec. 8.4.

Finally the CHT solver has been validated considering both a flat plate case and a nozzle guide vane flow. The results of the flat plate calculations have been compared with Fluent CHT results while for the NGV case, experimental data on the NASA-

C3X cooled nozzle guide vane (Hylton et al. (1983)) have been used as a reference. The results of the CHT validation are presented in Sec. 8.5.

Before discussing the results of the validations tests, the various boundary conditions and computational domains used will be illustrated in the opening section of the chapter.

8.1 Computational domains and boundary conditions

In the following section the computational domains and boundary conditions used for all the validation cases considered in the present chapter will be discussed.

At first the domains used for the fluid-only and CHT validations (i.e. flat plate and C3X NGV) will be illustrated. This will be followed by the description of the hollow cylinder and the L-shaped beam cases, used for the solid solver validation tests.

A clarification on the computational domains used for the simulations presented in this chapter is deemed necessary at this point. While all the performed simulations refer to 2D cases (and will be treated as such in favor of clarity and brevity), the developed CHT solver can only use 3D grids, as illustrated in Sec. A.1. For this reason all the cases analyzed use a 3D mesh, obtained through prismatic extrusion of a 2D grid, using zero-gradient conditions (symmetry and adiabatic for the fluid and solid domains respectively) along the boundaries normal to the extrusion direction. This allowed to model a 2D flow using a 3D domain. All final results have been checked to exhibit no gradients in the extrusion direction.

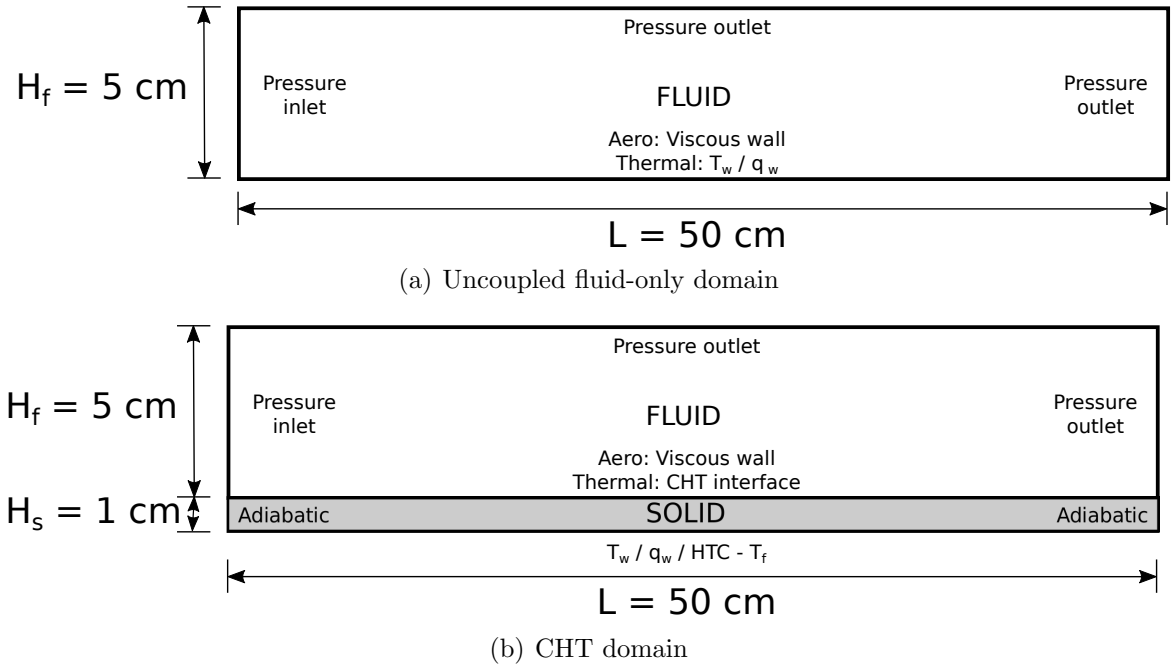


Figure 8.1: Computational domains and boundary types for the uncoupled and CHT Hydra flat plate calculations.

8.1.1 Fluid and CHT studies

8.1.1.1 Flat plate

Flat plate calculations have been run both on the original Hydra fluid solver and the implemented CHT solver. Fig. 8.1 shows the computational domains and the type of boundaries used in both cases.

Table 8.1 shows the values of the boundary conditions for the various cases analyzed.

Two flow conditions, laminar and turbulent, have been considered with $Re_L = 5 \times 10^4$ for the laminar case and $Re_L = 5 \times 10^5$ for the turbulent case. As shown in Table 8.1, two different values for the inlet total pressure have been used for laminar and turbulent simulations respectively, while the outlet static pressure and inlet total temperature have not been altered.

Table 8.1 reports also the imposed values of wall temperature, heat flux, HTC and

Aero Boundary Conditions			
Laminar	$p_{0in} = 101326.5 [Pa]$	$T_{0in} = 300 [K]$	$p_{out} = 101325 [Pa]$
Turbulent	$p_{0in} = 101470.1 [Pa]$		
Wall Thermal Boundary Conditions			
	Constant T_w	Constant $\dot{q}_w$	Convective BC
Laminar	310 [K]	75 [W/m ²]	$h = 10 [W/(m^2K)]$ $T_f = 320 [K]$
Turbulent	310 [K]	300 [W/m ²]	$h = 40 [W/(m^2K)]$ $T_f = 320 [K]$

Table 8.1: Boundary conditions used for Hydra flat plate calculations. The thermal boundary conditions are applied on the flat plate wall for uncoupled simulations and to the plate's bottom wall for CHT cases.

driving temperature, for the three kind of thermal boundary conditions considered on the flat plate case.

A clarification should be made concerning the thermal boundary conditions reported in Table 8.1. For the uncoupled fluid-only simulations (Fig. 8.1(a)) such conditions have been applied to the flat plate wall, while for the CHT computations (Fig. 8.1(b)) the flat plate wall was treated as a CHT interface and the same thermal boundary conditions were applied to the bottom wall of the flat plate solid domain. The results of fluid-only flat plate validations tests are discussed in Sec. 8.3 while the CHT flat plate validation is illustrated in Sec. 8.5.1, 8.5.2 and 8.5.3.

8.1.1.2 C3X Nozzle Guide Vane

To validate the developed conjugate solver on a typical NGV flow, the NASA-C3X cooled vane has been used as a benchmark. The vane is cooled using ten straight cylindrical cooling holes. Experimental data from linear cascade tests available from Hylton et al. (1983) have been used to validate the solver. The simulations have been carried out on the midspan section of the cascade geometry.

Conjugate calculations have been performed on the computational domain of

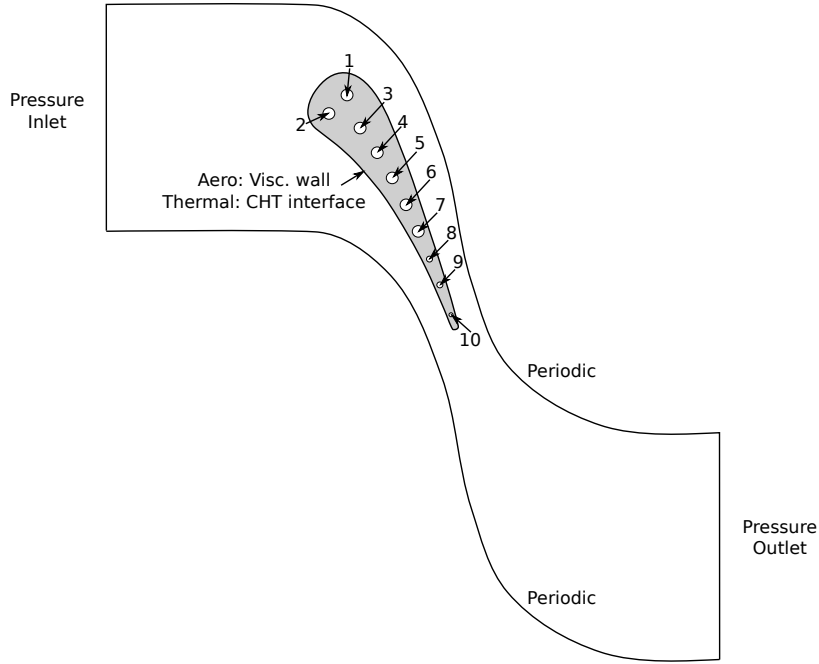


Figure 8.2: Computational domain and boundary types for the CHT simulations on C3X NGV.

M_{out}	0.9 [–]
Re_{out}	1.5×10^6 [–]
Tu_{in}	6.5% [–]
TR	0.7 [–]

Table 8.2: Flow conditions for the C3X experimental case No. 4311 (run 148) as from Hylton et al. (1983).

Fig. 8.2 considering the flow conditions shown in Table 8.2, corresponding to experimental case No. 4311 in Hylton et al. (1983). The boundary conditions imposed at the pressure inlet and outlet are summarized in Table 8.3. The inlet value for the eddy viscosity ratio has been calculated considering the measured levels of Tu at inlet (see Table 8.2) and assuming, in absence of measured values, a value of 5.0×10^{-3} [m] for the turbulent length scale. This value for the turbulent length scale has been also assumed by Facchini et al. (2004) for the same geometry.

The wall surfaces of cooling holes have been treated as convective heat transfer boundaries, where HTC and the driving temperature T_f have been specified for each cooling hole. The measured values of the average temperature of the cooling air for

p_{0in}	$2.45 \times 10^5 [Pa]$
T_{0in}	$802.0 [K]$
μ_t/μ	$1.0 \times 10^3 [-]$
p_{out}	$1.43 \times 10^5 [Pa]$

Table 8.3: Boundary conditions imposed at inlet and outlet for the C3X validation case.

Hole #	$D [mm]$	$C_r [-]$	$Re_D \times 10^5 [-]$	$Nu_D [-]$	$HTC [W/(m^2K)]$	$T_f [K]$
1	6.30	1.118	1.55	292.1	1412.0	360.63
2	6.30	1.118	1.62	302.1	1460.5	360.63
3	6.30	1.118	1.60	299.3	1404.0	346.23
4	6.30	1.118	1.59	297.9	1415.9	352.50
5	6.30	1.118	1.71	315.7	1465.3	341.41
6	6.30	1.118	1.62	302.1	1517.3	380.08
7	6.30	1.118	1.62	303.2	1441.2	352.47
8	3.10	1.056	1.04	201.1	2080.2	387.16
9	3.10	1.056	0.64	135.9	1495.1	421.81
10	1.98	1.025	0.47	102.9	1901.3	466.79

Table 8.4: Boundary conditions imposed at cooling holes for the C3X validation case. The cooling holes numbers refer to Fig. 8.2.

each cooling hole have been used as T_f : these values are reported in Table 8.4. The heat transfer coefficient for each cooling hole has been calculated using:

$$Nu_D = 0.022 C_r Pr^{0.5} Re_D^{0.8} \quad (8.1)$$

where Nu_D and Re_D are the diameter-based Nusselt and Reynolds numbers respectively, and C_r is a constant accounting for entry effects in each of the cooling holes. Values of Re_D and C_r are available from experimental measurements and are reported in Table 8.4 together with the corresponding values of Nu_D and HTC. The value of the Prandtl number has been assumed as 0.7.

The solid domain has been modeled as made of ASTM 310 stainless steel as the material used for the reference experimental test case. Unfortunately Hylton et al.

ρ	7900 [kg/m ³]
C	586.2 [J/(kg K)]
k	18.1 [W/(m K)]

Table 8.5: Thermal properties of ASTM 310 from Facchini et al. (2004).

(1983) did not report any measured values of the thermal properties of the solid for the particular blade used in the experiments. The used material properties are listed in Table 8.5.

The results of the C3X NGV validation study are discussed in Sec. 8.5.4.

8.1.2 Solid conduction studies

8.1.2.1 Hollow cylinder

The first validation test performed on the solid conduction solver considered a hollow cylinder subject to a fixed temperature gradient. The computational domain and boundary conditions used are shown in Fig. 8.3.

The results of Hydra calculation have been compared with the analytical solution for the same problem as well as with the results obtained using the FEM solver ANSYS Mechanical. The results of this validation test are discussed in Sec. 8.4.1.

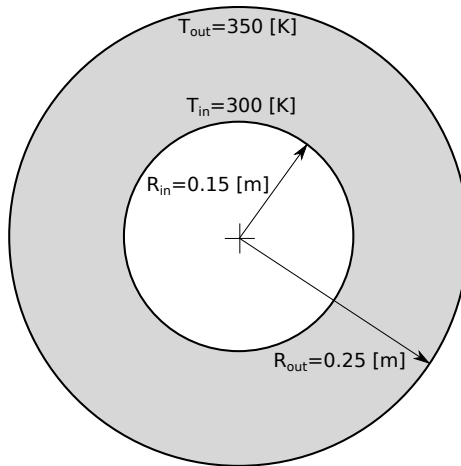


Figure 8.3: Computational domain and boundary conditions for the solid simulations on the hollow cylinder case.

8.1.2.2 L-shaped beam

A second validation test for the solid conduction solver has been performed on a L-shaped beam subject to a temperature gradient. The computational domain and boundary conditions used are shown in Fig. 8.4. Hydra results on this geometry

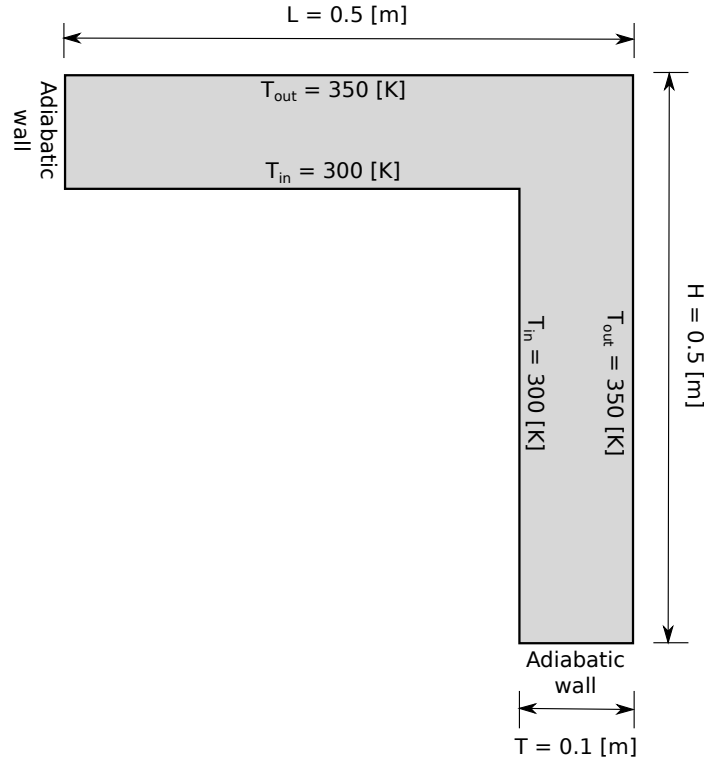


Figure 8.4: Computational domain and boundary conditions for the solid simulations on the L-shaped beam case.

have been compared against ANSYS Mechanical FEM solution. The results of this validation test are discussed in Sec. 8.4.2.

8.2 Preliminary Hydra fluid-only validation

The results of the validation of the existing Hydra flow solver for flat plate boundary layer flow are reported in this section.

The grid used for the fluid-only calculations is the same grid used in the fluid domain for the conjugate flat plate simulations (Fig. 8.13). The fluid domain has been

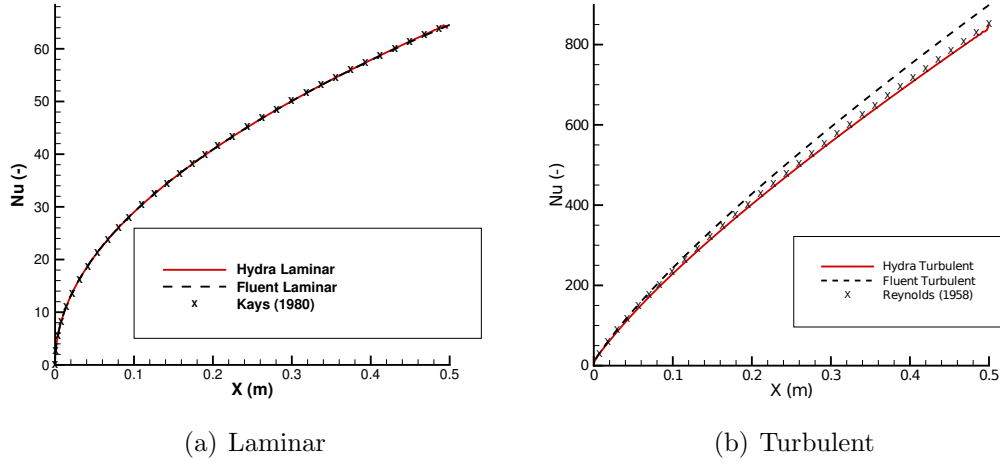


Figure 8.5: Uncoupled calculations on laminar and turbulent boundary layer for constant T_w boundary condition: Nu comparison between Hydra, Fluent and the reference correlation.

discretized with around 1.6×10^4 nodes. The mesh resolution is of the same order than the one used for the previously discussed Fluent flat plate calculations (Sec. 3.3) and has been considered to be adequate for the scope of the simulation. Wall spacing was chosen to obtain $y^+ < 1$ on the whole wall length. Following the same procedure of Fluent validation tests on flat plate flow, the wall to mainstream temperature difference has been kept as low as 10 [K] to reduce the effects of compressibility due to temperature gradients.

The results of the performed calculations are shown in Fig. 8.5 for both laminar and turbulent flow conditions. As done for Fluent flat plate validation, laminar results have been compared with Eqn. (3.3) (see Kays & Crawford (1980)), while Eqn. (3.4) (Reynolds et al. (1958)) has been used as a reference for the turbulent case. Fluent results under the same flow conditions have been also included as a benchmark.

Turbulent simulations have been performed only considering Spalart Allmaras turbulence model as it was considered adequate enough for the cases considered and it was of no interest at this point to study the effect of turbulence models on the solution.

The results of Fig. 8.5 show a very good agreement between Hydra results and the reference correlations for both laminar and turbulent cases. For the turbulent case it can be observed that Hydra gives a better accuracy compared with Fluent with respect to the correlation of Eqn. (3.4). It can be also observed from Fig. 8.5(b) that Hydra tends to underestimate the heat transfer levels for turbulent flow with respect to Fluent. The differences in the turbulent predictions are to be attributed to different implementation of the turbulence model between Hydra and Fluent.

As a final remark the results shown confirm a successful validation of Hydra flow solver for a flat plate flow under isothermal wall conditions. This validation allowed to assess the accuracy of the original solver and to evaluate the quality of the solution obtained using the newly implemented thermal boundary conditions, to be discussed in the following section.

8.3 Validation of implemented thermal boundary conditions

8.3.1 Heat flux boundary condition

The same computational domain, grid and flow conditions of the previously shown flat plate results, have been used to validate the developed wall heat flux boundary condition described in Sec. 7.5.2. Also in this case both laminar and turbulent flow regimes have been considered. As shown in Table 8.1 a constant heat flux of $75 [W/m^2]$ has been applied to the flat plate wall for the laminar simulations, while $\dot{q}_w = 300 [W/m^2]$ for the turbulent case.

The reference wall Nu has been calculated using Eqn. (8.2) for laminar flow and Eqn. (8.3) for turbulent flow (Kays & Crawford (1980)).

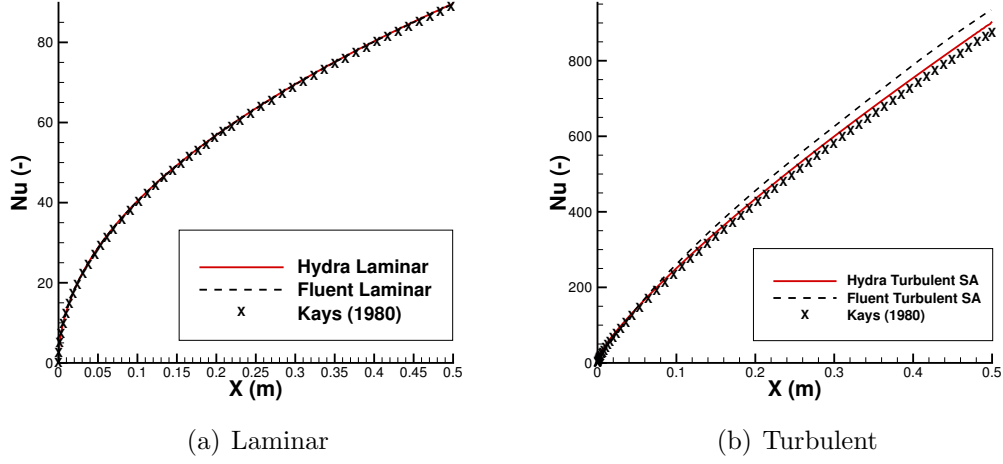


Figure 8.6: Uncoupled calculations on laminar and turbulent boundary layer for constant $\dot{q}_w$ boundary condition: Nu comparison between Hydra, Fluent and the reference correlation.

$$Nu_x = 0.453Pr^{1/3}Re_x^{1/2} \quad (8.2)$$

$$Nu_x = 0.030Pr^{0.6}Re_x^{0.8} \quad (8.3)$$

The results of the validation test are shown in Fig. 8.6. Hydra simulations are in very good agreement with the reference correlations in both laminar and turbulent cases, indicating a successful validation of the implemented boundary condition. Furthermore by comparing Fig. 8.5 and Fig. 8.6 it can be seen that Hydra predictions show similar discrepancy with the reference correlations under both constant temperature and constant heat flux conditions: this allows to conclude that the implemented wall boundary condition did not have a negative effect on the accuracy of the solver. It is also worth mentioning that no change in the convergence rate of the solver has been observed when using the implemented constant heat flux boundary condition.

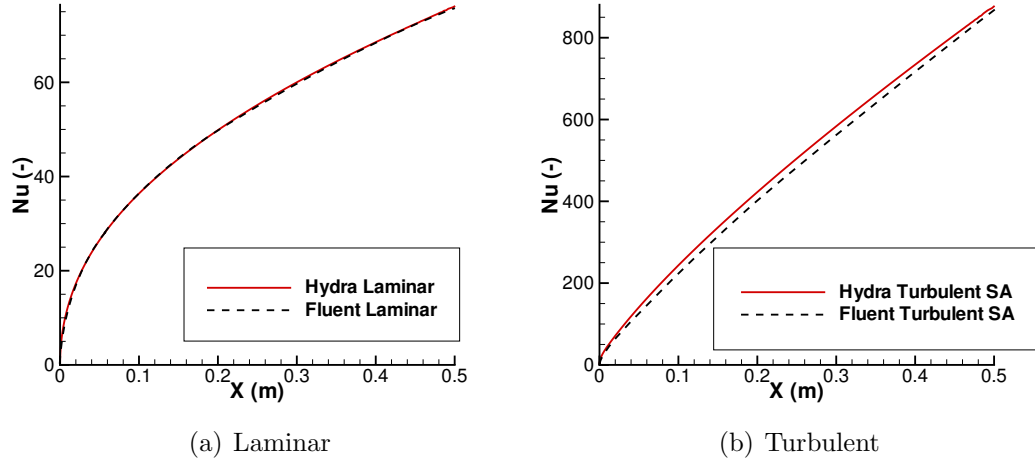


Figure 8.7: Uncoupled calculations on laminar and turbulent boundary layer for convective heat flux boundary condition: Nu comparison between Hydra and Fluent.

8.3.2 Convective heat flux thermal boundary condition

According to the author's knowledge, no experimental correlation, nor analytical result is available to establish a reference case for flat plate flow under the convective heat transfer conditions for given HTC and T_f . For this reason Hydra results on flat plate using the implemented convective heat flux boundary condition have been compared only with Fluent calculations.

As shown in Table 8.1 a wall heat transfer coefficient of $10 [W/(m^2 K)]$ and a fluid driving temperature $T_f = 320 [K]$ has been applied to the flat plate wall for the laminar simulations, while $h = 40 [W/(m^2 K)]$ and $T_f = 320 [K]$ for the turbulent case.

The results of the CFD calculations are shown in Fig. 8.7. As was the case for the constant heat flux cases, globally a good agreement between Hydra and Fluent results can be seen, with the results of the laminar simulations showing an almost perfect match. It is worth noting how the difference between Hydra and Fluent Nu for the convective heat flux boundary condition reaches a maximum around $1/3$ of the plate to decrease further downstream. This is different to what observed for the constant

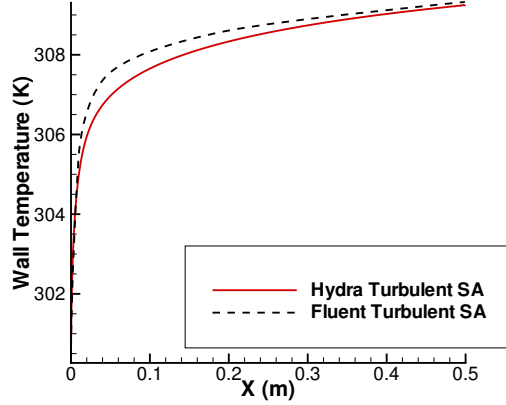


Figure 8.8: Uncoupled calculations turbulent boundary layer for convective heat flux boundary condition: T_w comparison between Hydra and Fluent.

heat flux boundary conditions where the differences were increasing monotonically with X . This different behavior can be easily understood from the following.

Consider an iso-heat flux simulation where $\dot{q}_w > 0$ (heating of the flow) is imposed at the wall. A lower Nu on the fluid side corresponds to a higher thermal resistance of the flow, that will be picking up less heat for unit length, being $\dot{q}_w$ constant. This results in a cooler flow and this effect is additional, until thermal equilibrium is reached (fully developed thermal boundary layer).

When imposing a convective boundary condition the wall heat flux becomes function of the same wall temperature. Figure 8.8 shows the wall temperature distribution for Hydra and Fluent flat plate calculation using the Robin thermal boundary condition. In both cases the wall temperature at the start is practically the inlet total temperature. This is due to the extremely high values of flow HTC at the flat plate leading edge. Hydra solution evolves with a cooler wall due to the higher thermal resistance of the flow but this corresponds also to higher level of wall heat transfer compared to a hotter wall case. The increased heat flux level counteracts the higher flow thermal resistance and is the cause of the observed trends in the differences in T_w and Nu for Hydra and Fluent solutions under convective heat transfer boundary condition.

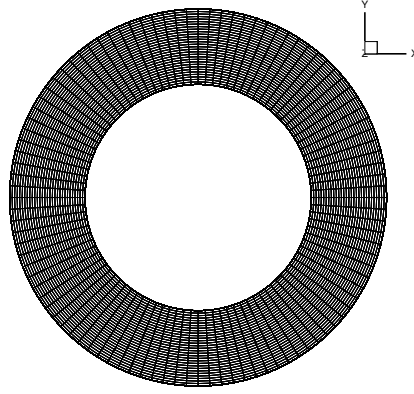


Figure 8.9: Grid used for the cylinder solid conduction problem.

8.4 Solid solver validation

This section will illustrate the results of the two validation test performed on the solid solver: the hollow cylinder and the L-shaped beam.

8.4.1 Hollow cylinder

For the hollow cylinder test case the computational domain of Fig. 8.3 has been discretized with 30 radial nodes. The computational mesh used is shown in Fig. 8.9.

Considering steady conduction in the solid, the analytical solution of the heat transfer equation on the considered domain reads:

$$T(r) = A \ln(r) + B \quad (8.4)$$

with $A = 97.88$ and $B = 485.69$ considering the boundary conditions for the specific case considered (shown in Fig. 8.3).

The validation of the solver has been performed comparing Hydra results with the above mentioned analytical solution as well as with the Finite Element solution obtained using ANSYS Mechanical. Figure 8.10 shows the result of such comparison. A perfect agreement can be observed between Hydra and the two benchmark cases considered.

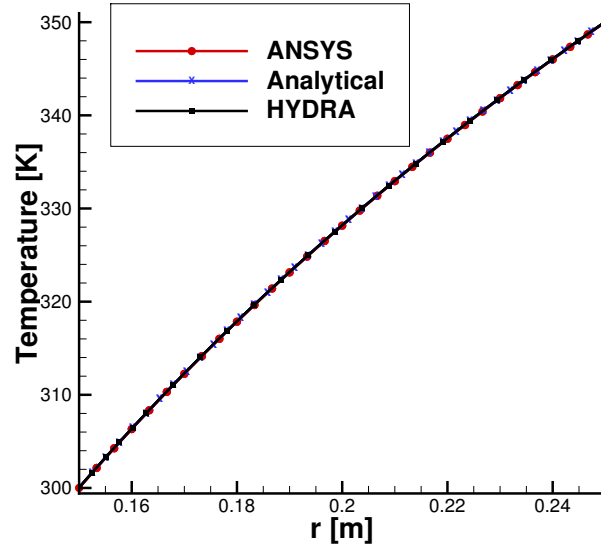


Figure 8.10: Temperature distribution in the cylinder: Hydra solid solver, ANSYS Mechanical and analytical solution.

8.4.2 L-shaped beam

For the L-shaped beam case, the computational domain of Fig. 8.4 has been discretized with 2000 nodes. The resulting grid is shown in Fig. 8.11.

For this case Hydra results have been only compared with ANSYS Mechanical solution. The results of this validation test, showing the solid temperature contours near the corner region are shown in Fig. 8.12. A perfect agreement between the two solutions can be observed, concluding successfully the validation of the implemented Hydra solid solver.

8.5 Conjugate Heat Transfer solver validation

In this section the CHT validation tests performed on the flat plate domain will be discussed. Particularly in Sec. 8.5.1 the validation tests performed using an isothermal condition on the flat plate bottom surface will be illustrated, followed by the iso-heat flux cases (Sec. 8.5.2) and the convective heat flux cases (Sec. 8.5.3). Fluent CHT

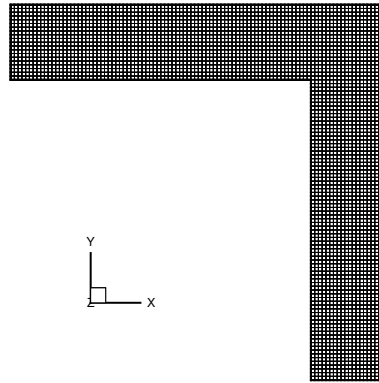


Figure 8.11: Grid used for the corner solid conduction problem.

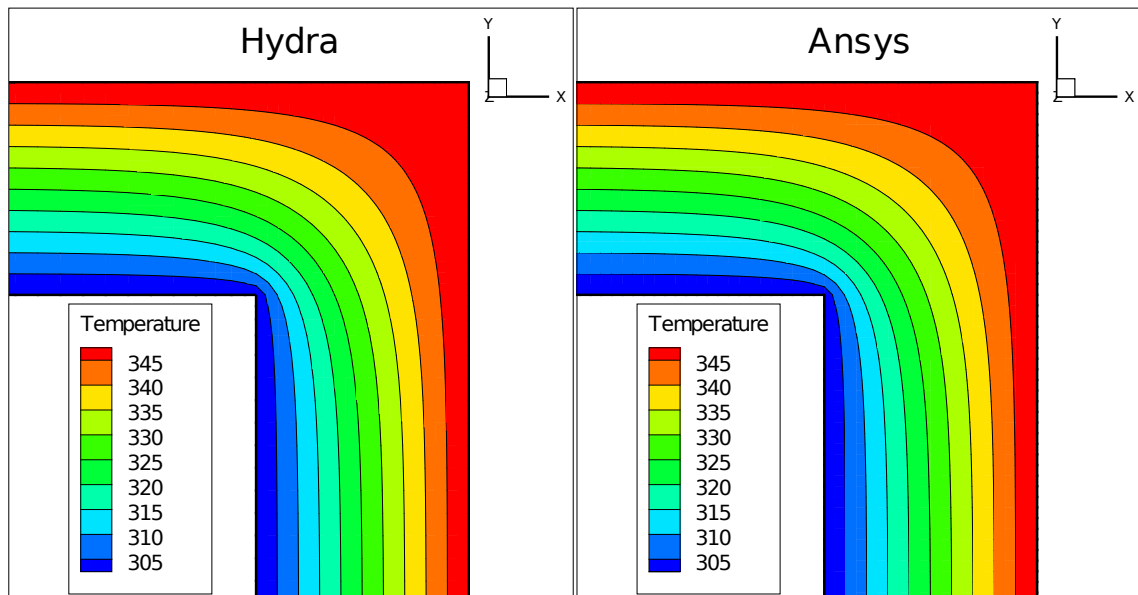
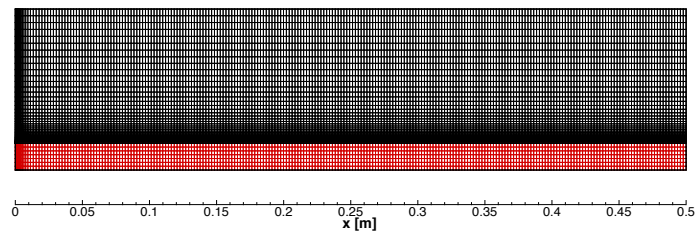


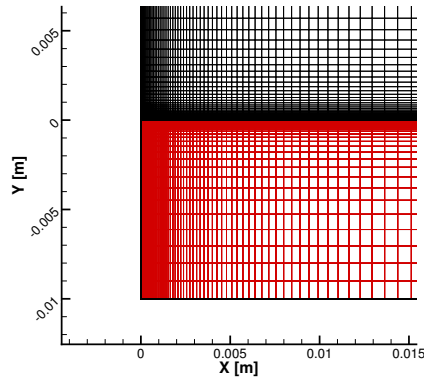
Figure 8.12: Comparison of temperature contours in the corner region of the L-shaped beam: Hydra solid solver and ANSYS Mechanical.

simulations will be used as a benchmark to compare with Hydra results in all the considered cases.

All calculations are performed using the boundary conditions of Table 8.1. The computational domain, shown in Fig. 8.1(b), has been discretized with 15000 nodes for the fluid domain and 7500 nodes in the solid domain. A global view and a detail of the mesh in the leading edge area are shown in Fig. 8.13.



(a) Full grid



(b) Detail

Figure 8.13: Grid used for the flat plate CHT calculations.

8.5.1 Flat plate with constant temperature at the bottom wall

8.5.1.1 Laminar case

The comparison between Hydra CHT and Fluent CHT simulation on laminar flow on flat plate is shown in Fig. 8.14. The figure shows the temperature contours in the fluid and solid domains for the whole extension of the boundary layer (Detail-1) as well as for a close up in the leading edge region (Detail-2), where most of the thermal gradient in the solid is present. When looking at Fig. 8.14 the reader should note that the contour plot scales are different for the Detail-1 and Detail-2 regions.

The agreement between Hydra results and Fluent results is very good for the considered case with both simulations resulting in an almost identical thermal field both in the fluid and solid domains.

As a further comparison of Hydra and Fluent results, Fig. 8.15 shows the wall temperature distribution for both solvers. As expected from the previous results a very good agreement is found.

For this case only a further investigation on Hydra CHT results is performed to check that the implemented CHT solver does actually enforce continuity of wall temperature and heat flux at the fluid-solid interface. Figures 8.16 and 8.17 show respectively the absolute value of the wall heat flux and the wall temperature as seen from the fluid-side and solid-side of the CHT wall. The absolute value of the heat flux is taken as a positive heat flux for the solid side is seen as negative by the fluid side and vice versa. The results confirm that continuity of temperature and heat flux is retained at the CHT interface.

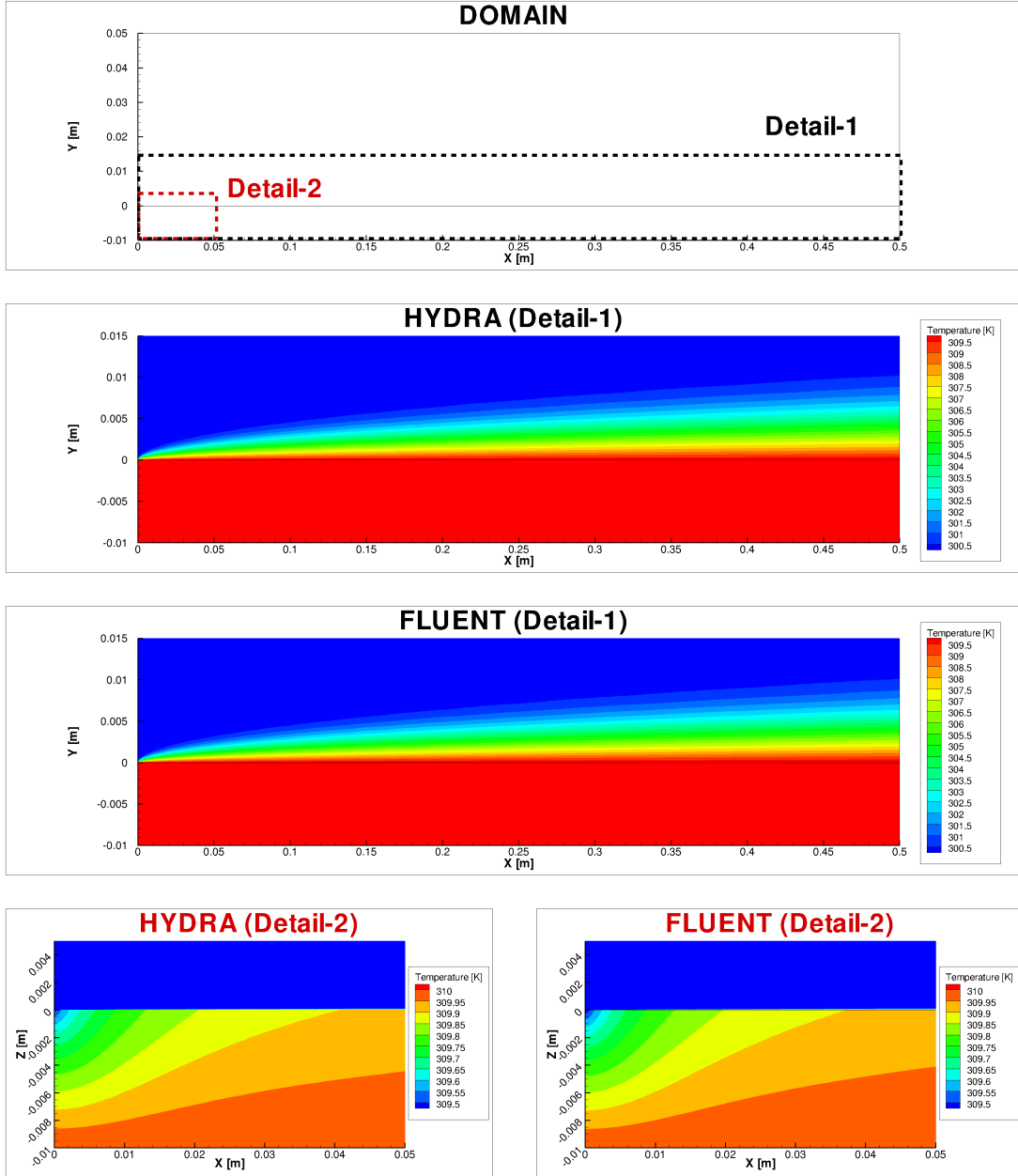


Figure 8.14: Flat plate CHT calculation, constant T_w case, laminar flow, thermal field in both the fluid and solid domain: comparison between Hydra CHT and Fluent.

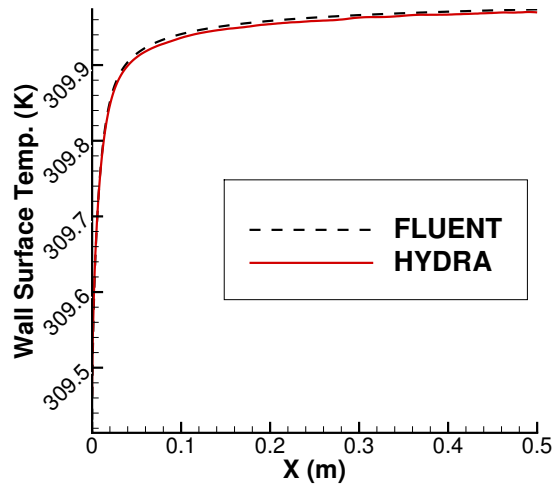


Figure 8.15: Flat plate CHT calculation, constant T_w case, laminar flow: flat plate T_w comparison between Hydra CHT and Fluent.

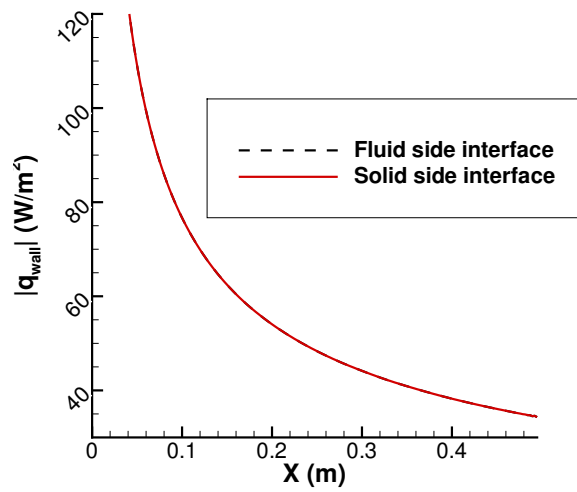


Figure 8.16: Continuity of wall heat flux at fluid and solid side of the CHT interface.

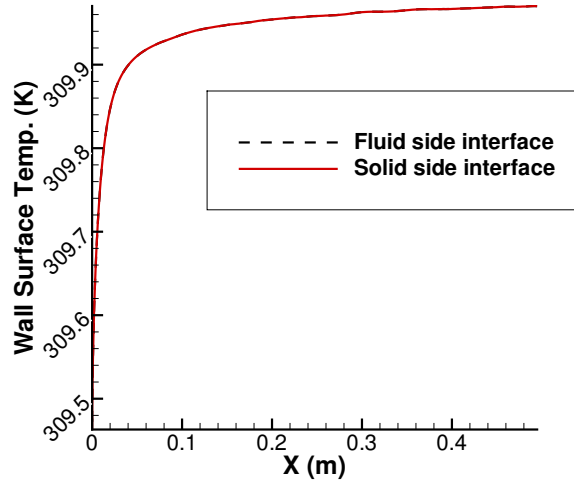


Figure 8.17: Continuity of wall temperature at fluid and solid side of the CHT interface.

8.5.1.2 Turbulent case

The results of the turbulent flat plate calculations with constant wall temperature on the bottom of the flat plate wall are shown in Fig. 8.18. Also in this case a close up to the leading edge of the flat plate (Detail-2) has been performed to highlight the temperature field in the region.

The comparison of the wall temperature distribution predicted by the two codes is shown in Fig. 8.19. It can be observed that Hydra results predict a marginally warmer wall temperature than Fluent. This is consistent with the lower Nu predicted by Hydra for turbulent simulations (see Fig. 8.5(b)).

As was for the laminar case previously shown, a very good agreement can be observed between the implemented solver and Fluent: this applies to both the fluid and solid thermal fields as well as the predicted wall temperature distribution.

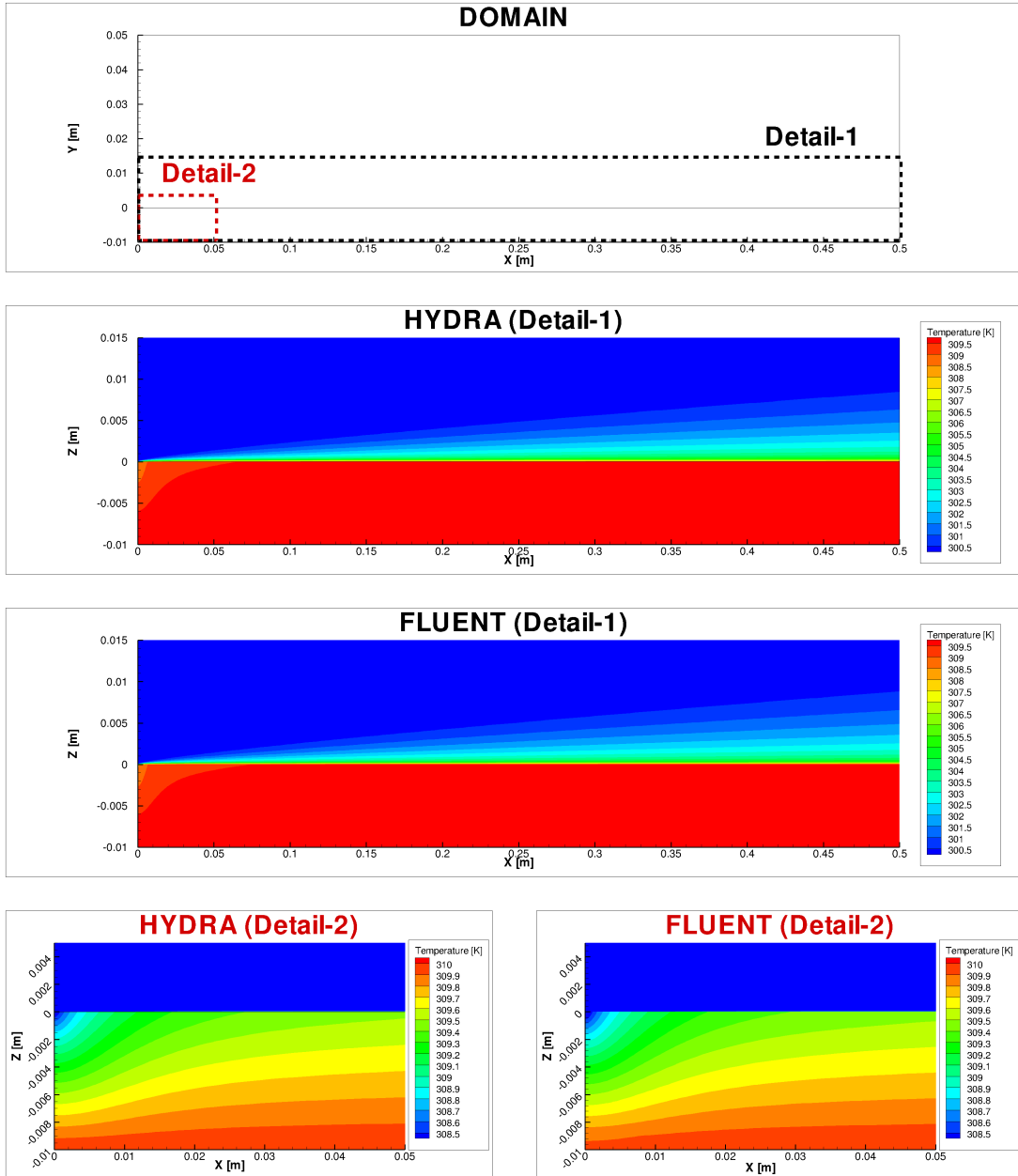


Figure 8.18: Flat plate CHT calculation, constant T_w case, turbulent flow, thermal field in both the fluid and solid domain: comparison between Hydra CHT and Fluent.

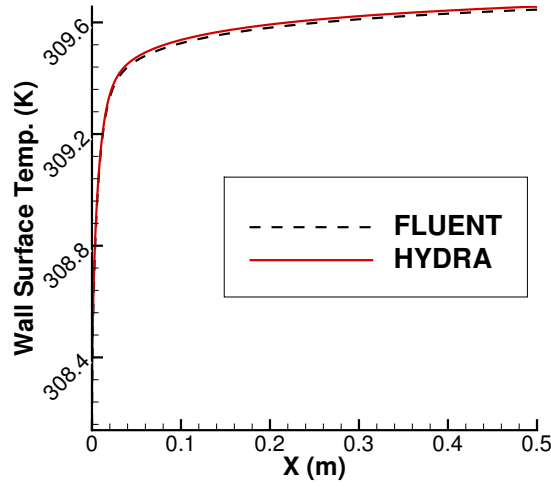


Figure 8.19: Flat plate CHT calculation, constant T_w case, turbulent flow: flat plate T_w comparison between Hydra CHT and Fluent.

8.5.2 Flat plate with constant heat flux at the bottom wall

8.5.2.1 Laminar case

Figure 8.20 shows the thermal field in the fluid and solid domains for the laminar flow case. Differently from the constant temperature simulations, the temperature gradient in the solid domain spreads throughout the length of the flat plate for the analyzed constant heat flux condition. The resulting wall temperature distribution is shown in Fig. 8.21.

Also for this case it can be concluded that Hydra results are in very good agreement with Fluent predictions.

8.5.2.2 Turbulent case

The results of the turbulent case for imposed heat flux on the bottom of the flat plate are shown in Fig. 8.22, illustrating the thermal fields in the solid and fluid domain as predicted by Hydra CHT solver and Fluent. The wall temperature distribution is shown in Fig. 8.23.

A very close match between Hydra and Fluent predictions can be observed also

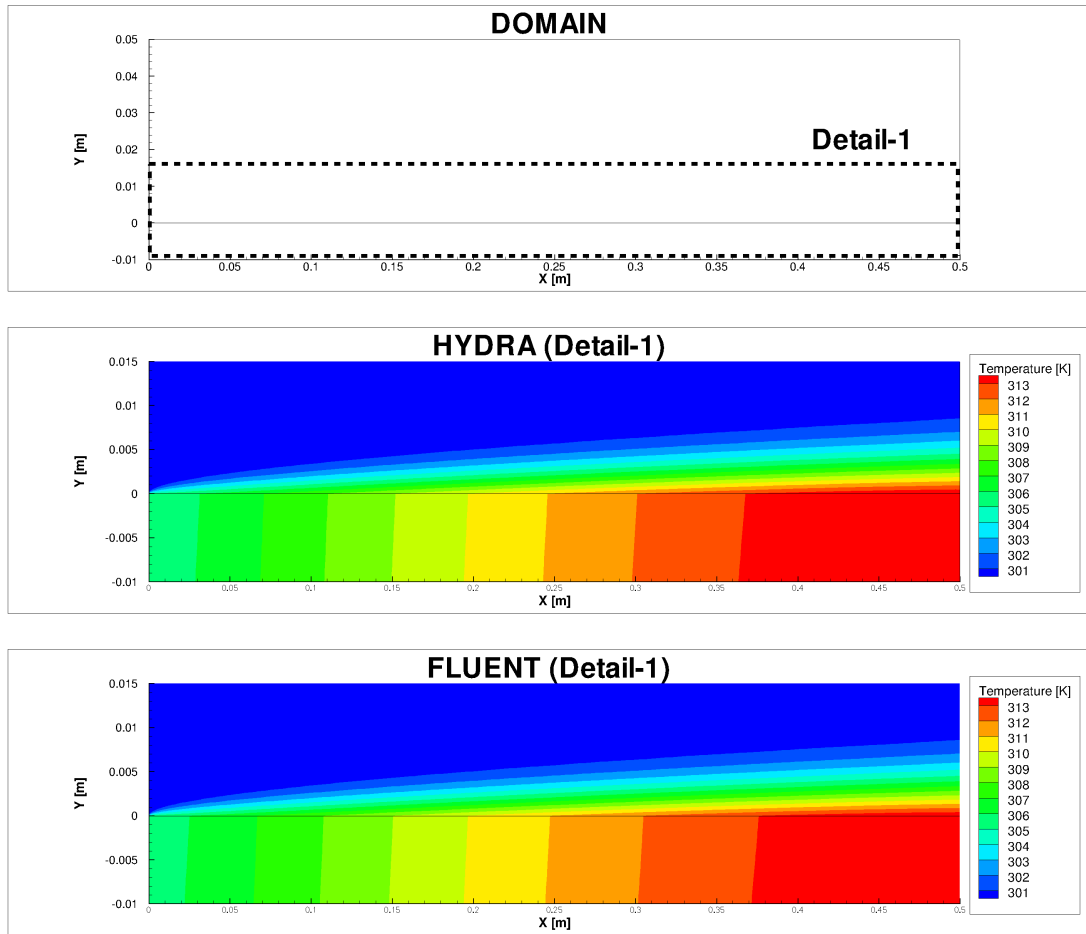


Figure 8.20: Flat plate CHT calculation, constant $\dot{q}_w$ case, laminar flow, thermal field in both the fluid and solid domain: comparison between Hydra CHT and Fluent.

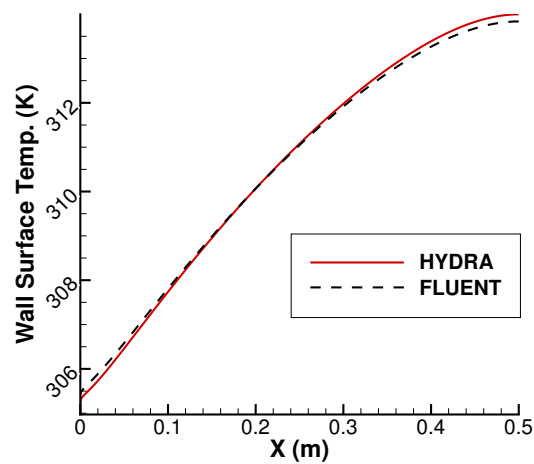


Figure 8.21: Flat plate CHT calculations for the constant $\dot{q}_w$ case: laminar flow. Wall temperature.

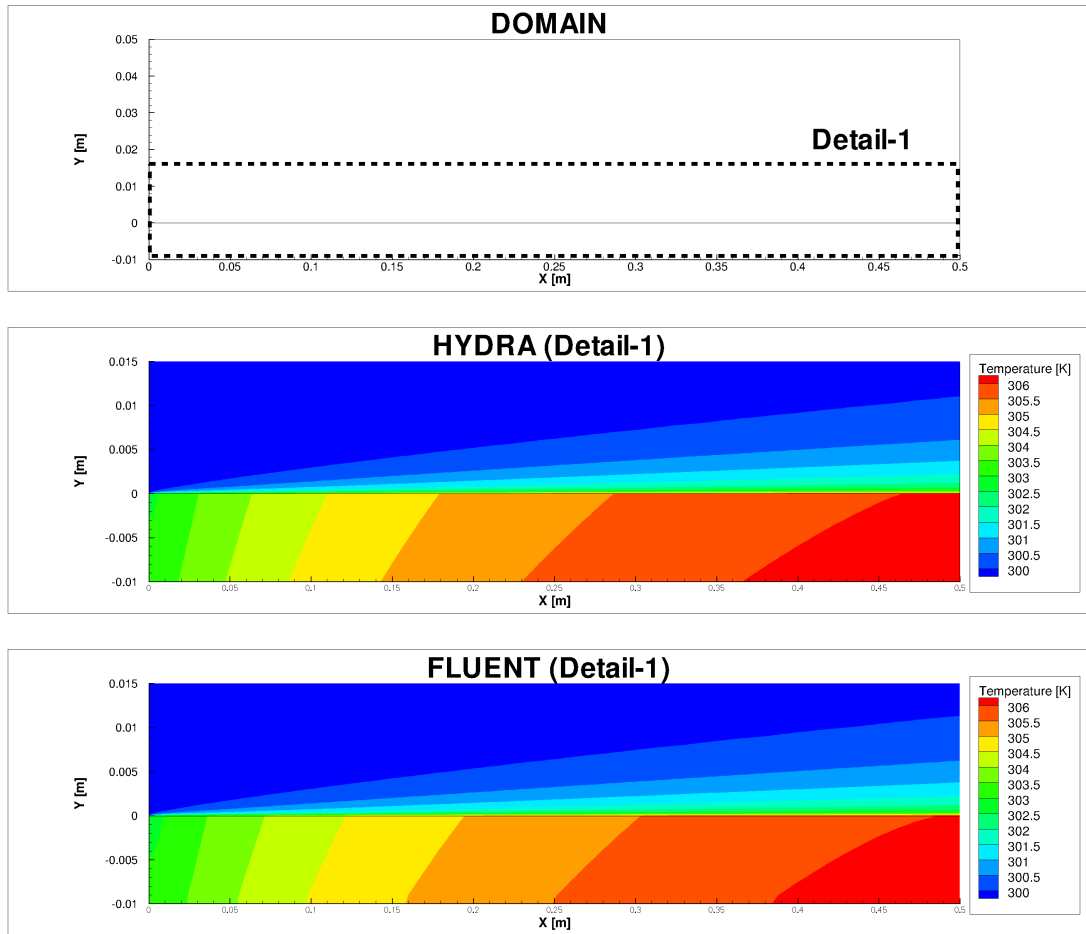


Figure 8.22: Flat plate CHT calculation, constant $\dot{q}_w$ case, turbulent flow, thermal field in both the fluid and solid domain: comparison between Hydra CHT and Fluent.

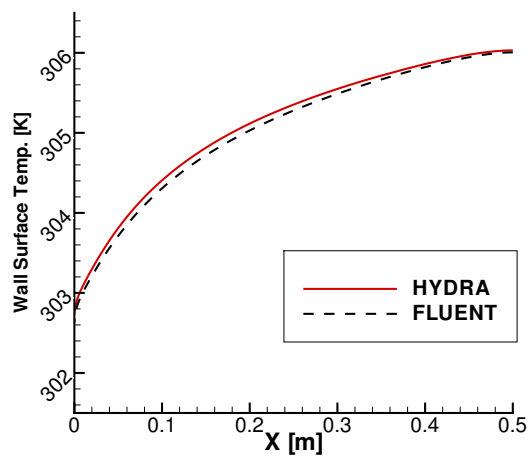


Figure 8.23: Flat plate CHT calculations for the constant $\dot{q}_w$ case: turbulent flow. Wall temperature.

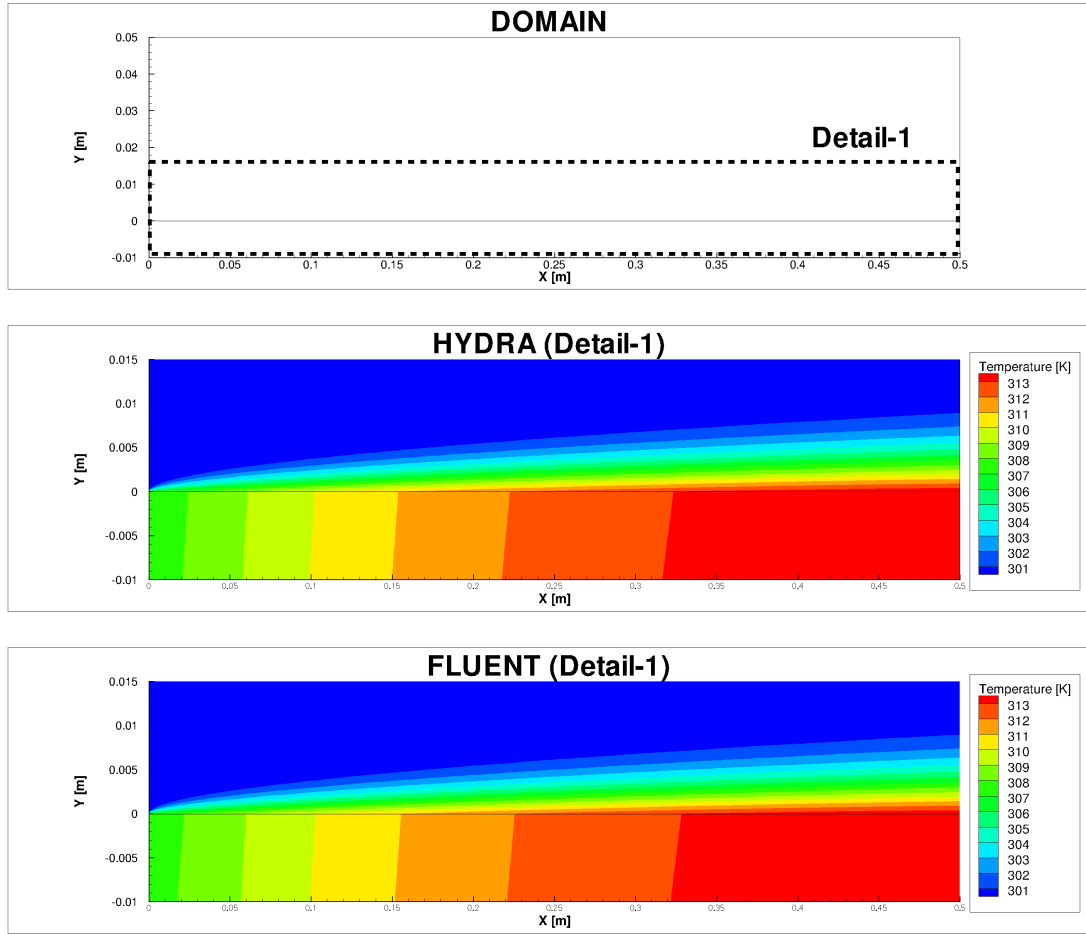


Figure 8.24: Flat plate CHT calculation, convective boundary condition case, laminar flow, thermal field in both the fluid and solid domain: comparison between Hydra CHT and Fluent.

for this case.

8.5.3 Flat plate with convective heat flux

8.5.3.1 Laminar case

Figure 8.25 shows the comparison between the thermal field in both fluid and solid domains as predicted by Hydra and Fluent CHT calculations. The resulting wall temperature distribution are plotted in Fig. 8.25. The two solvers show very similar temperature predictions in both cases considered, similarly to the previously analyzed cases.

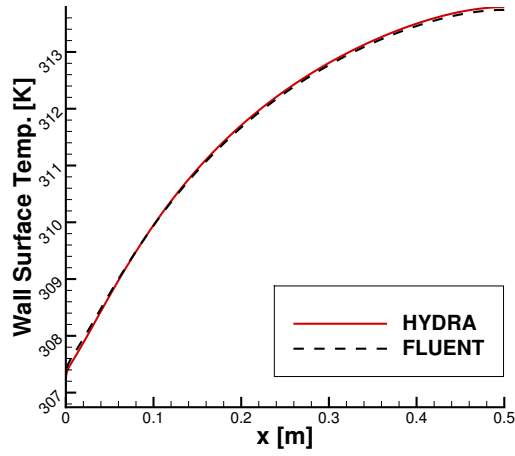


Figure 8.25: Flat plate CHT calculations for the convective heat flux case: laminar flow. Wall temperature.

8.5.3.2 Turbulent case

Finally, in Fig. 8.26 are shown the temperature contours in the flat plate and external boundary layer in the case of turbulent flow when convective boundary condition is applied to the bottom wall of the plate. In Fig. 8.27 the corresponding wall temperature distribution is plotted.

Similarly to what was seen in the previously shown comparisons, a very good agreement between Hydra and Fluent is observed.

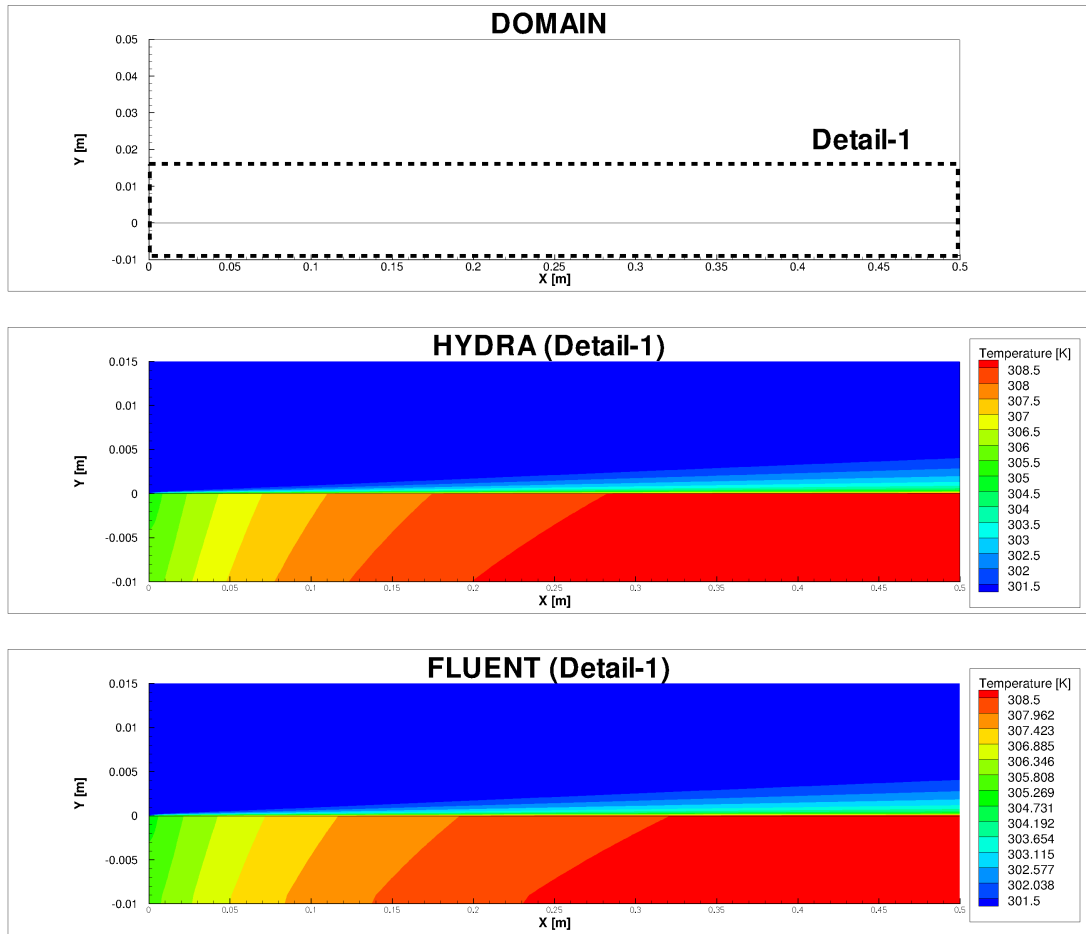


Figure 8.26: Flat plate CHT calculation, convective boundary condition case, turbulent flow, thermal field in both the fluid and solid domain: comparison between Hydra CHT and Fluent.

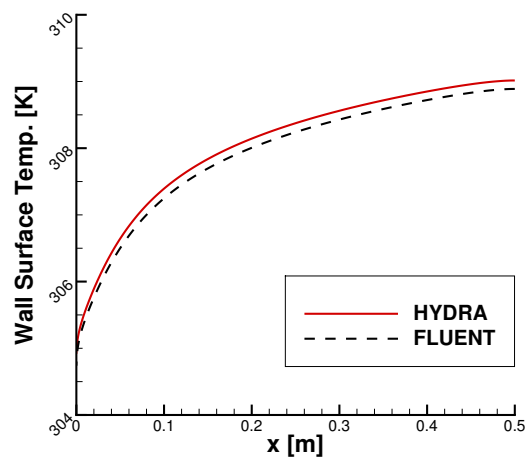


Figure 8.27: Flat plate CHT calculations for the convective heat flux case: turbulent flow. Wall temperature.

8.5.4 C3X nozzle guide vane

The computational domain and boundary conditions used for the C3X validation test have already been discussed in Sec. 8.1.1.2. This section does illustrate the results of the validation, together with the preliminary mesh dependency study performed.

8.5.4.1 Mesh dependency

Two grids have been considered for the mesh dependency study: a coarse mesh with 75×10^3 nodes (60×10^3 fluid and 15×10^3 solid nodes) and a fine mesh with a total of 125×10^3 nodes (100×10^3 for the fluid domain and 25×10^3 for the solid). Figure 8.28 shows a view of the coarser mesh used. It should be noted that the topology of the mesh is such that each cooling hole is surrounded by an O-mesh block, and the same solid-fluid interface is meshed with an O-grid on the fluid side (corresponding to the boundary layer O-grid) and one on the solid side. This allows to control the skewness of the near wall nodes on both sides of the CHT interface and on the cooling holes as well as to avoid the use of corner nodes (see Sec. 7.3.1) for this validation case.

Figure 8.29(a) shows the non-dimensional static pressure distribution on the profile using the two grids. Both mesh densities predict the same pressure distribution.

In Fig. 8.29(b) the wall non-dimensional HTC is shown. Such non-dimensional HTC is defined as:

$$\frac{HTC}{HTC_{ref}} = \frac{\dot{q}_w}{HTC_{ref}(T_{0in} - T_w)} \quad (8.5)$$

where $HTC_{ref} = 1135 [W/(m^2 K)]$. This representation of the wall heat flux has been chosen in line with the way experimental data used for the validation are reported in Hylton et al. (1983). The quantity on the abscissa should be intended as the distance from the leading edge (positive on the suction side, negative on the pressure side), non-dimensionalized by the pressure side and suction side lengths respectively. Both meshes used result in almost the same wall non-dimensional HTC distribution,

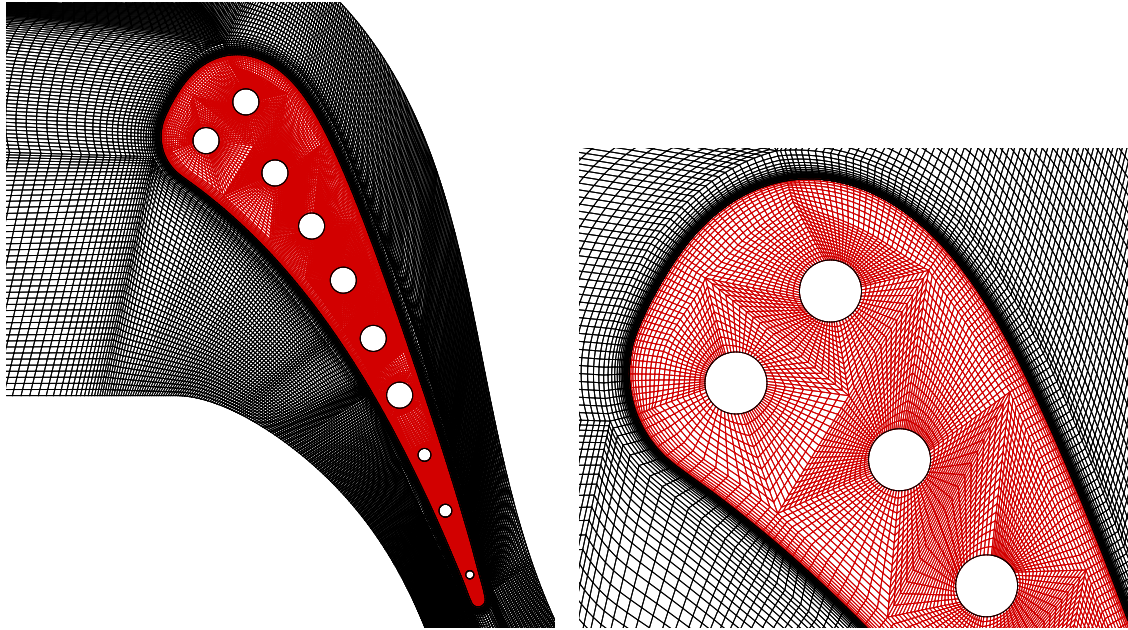


Figure 8.28: Mesh used for CHT calculation on C3X NGV.

indicating mesh independent results for the grid used.

It should be noted how the boundary layer on the C3X vane exhibits transitional behavior on the suction side. This is highlighted by Hylton et al. (1983) and clearly shown in the experimental results that will be presented in the next section. In order to capture such transitional behavior and predict correctly the heat load in the transitional region, the implemented transition modeling presented in Sec. 7.5.1 has been used. Figure 8.29(b) shows the extension of the laminar and transitional regions on the suction side. The transition point, extension of the laminar region and transition length have been chosen to match the observed boundary layer behavior.

Figure 8.30 shows the temperature distribution in the solid domain for both grids considered. The temperature contours are in very good agreement, confirming the grid independency of the considered results.

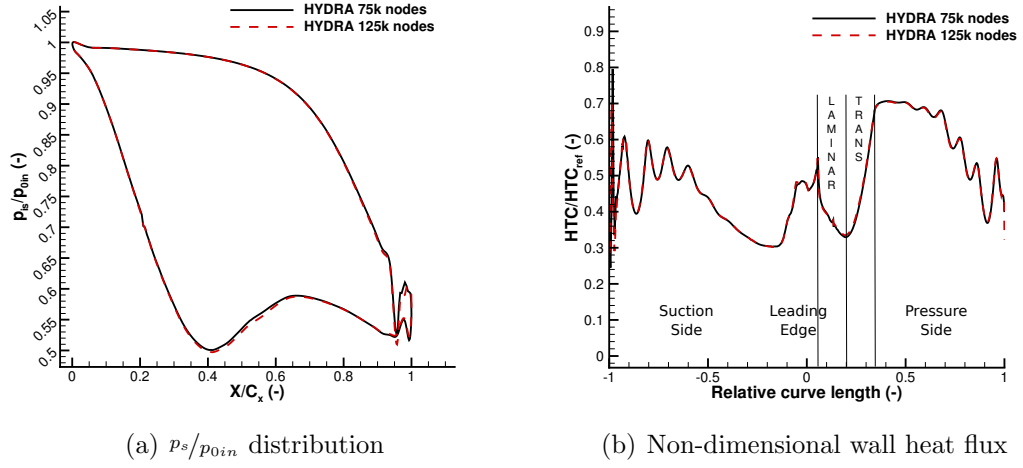


Figure 8.29: Mesh dependency study for C3X NGV: pressure distribution and wall non-dimensional HTC.

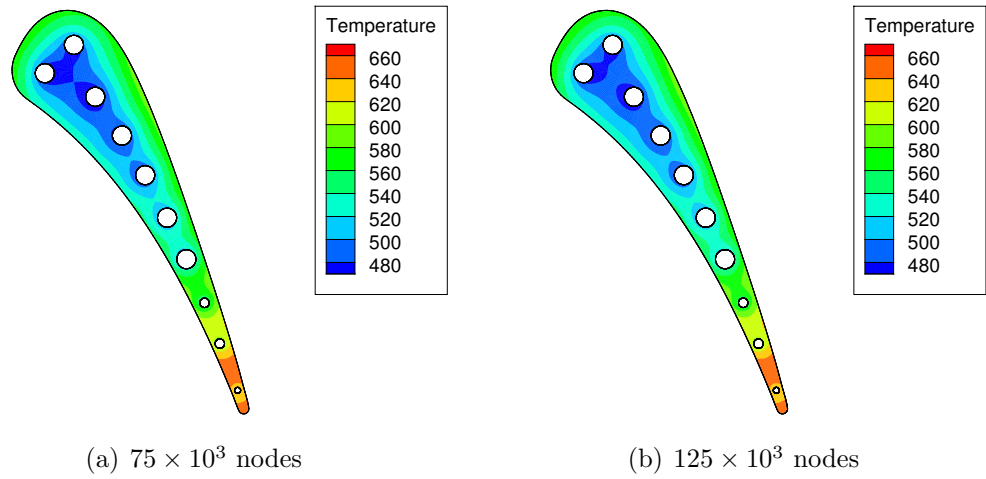


Figure 8.30: Mesh dependency study for C3X NGV: solid temperature distribution.

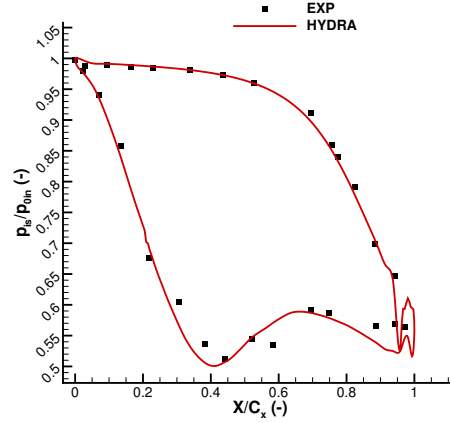


Figure 8.31: Results of the aerodynamic validation on the C3X NGV.

8.5.4.2 Validation results

In Fig. 8.31 the computed pressure distribution is compared with experimental data at midspan of the test cascade. No particular differences can be observed.

Before discussing the results of the thermal validation tests it is worth outlining the procedure used by Hylton et al. (1983) to obtain the wall heat flux from experimental data. The test cascade was only instrumented with thermocouples to obtain the distribution of wall temperature on the blade. A FEM model of the midspan profile was then generated and the thermal field in the solid was obtained through a numerical solution on a 200 nodes grid, imposing convective boundary condition on each of the cooling holes as described in Sec. 8.1.1.2. The wall heat flux was then obtained from the FEM solution and non-dimensionalized using the experimental wall temperature distribution. In the following discussion all references to *experimental* heat flux data have to be considered as being obtained through the above-described methodology.

In Fig. 8.32 the comparison between experimental non-dimensional HTC and Hydra predictions is shown. The experimental uncertainty on HTC values increases towards the trailing edge as the relative uncertainty on the geometrical definition of the blade increases with respect to the decrease of blade thickness (intended as distance from the cooling hole wall to the airfoil).

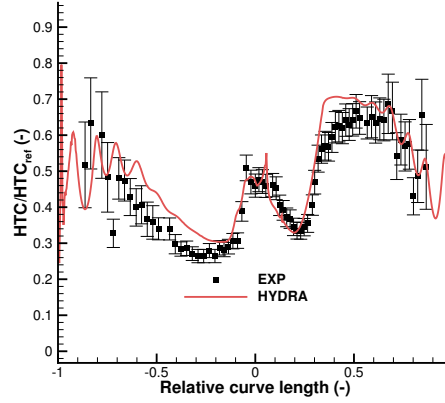


Figure 8.32: Results of the thermal validation on the C3X NGV: wall non-dimensional HTC.

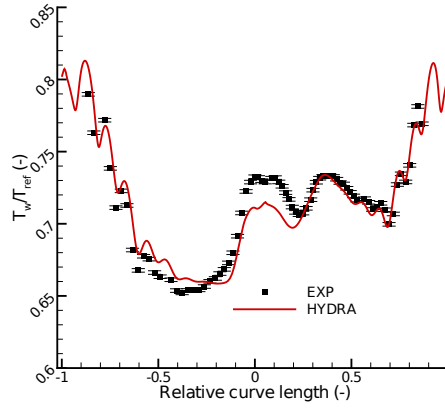


Figure 8.33: Results of the thermal validation on the C3X NGV: non-dimensional T_w .

Figure 8.33 shows the non-dimensional wall temperature distribution predicted by Hydra CHT computations and that measured experimentally. The uncertainties on the wall temperature measurements are estimated by Hylton et al. (1983) to be of the order of $\pm 1 [K]$.

Both the HTC and wall temperature distributions indicate the presence of transition on the suction side as explained previously.

The wall temperature distribution as predicted by Hydra is in very good agreement with experiments exception made for the stagnation point region where differences of about $15 [K]$ are seen between Hydra and experiments. Globally a satisfactory agreement is seen.

As far as HTC predictions is concerned (Fig. 8.32) Hydra results are in very good agreement in the leading edge region but tend to overestimate the heat transfer levels after the LE on both pressure side and suction side. The trends are nonetheless matched extremely well.

It is in the author beliefs that a main factor for the cited differences between Hydra results and experiments is the uncertainty on the material conductivity. The authors of the experimental report only mention an assumed uncertainty of 3% for the thermal conductivity of the material but give no indication of its average value. This is particularly vague considering, by assuming a typical variation of the thermal conductivity with temperature as proposed by Wlassow et al. (2010), variations of 15% of k can be observed in the range of temperature present in the investigated case.

Nonetheless, considering the relative numerical and experimental uncertainties, the results of Fig. 8.32 and Fig. 8.33 show that a reasonably good agreement is found between experimental data and Hydra CHT computations.

8.6 Final remarks

The validation tests performed on the implemented CHT solver have been presented. At first a preliminary validation of the existing Hydra flow solver has been performed on a simple flat plate case. This allowed to estimate the quality of the solution under the newly developed boundary conditions, ensuring that same levels of accuracy were retained when imposing heat flux or a convective boundary conditions at a solid wall.

The developed solid solver has been validated against analytical solution on a hollow cylinder as well as against FEM solution for a L-shaped beam. The solid solver showed a perfect match with the reference cases.

Hydra CHT solution for a boundary layer case has been obtained using all the three available thermal boundary conditions (constant temperature, constant heat

flux and convective heat flux, as implemented) for the bottom wall of the flat plate. Both laminar and turbulent flow conditions have been analyzed. Fluent CHT calculations have been taken as a reference for the considered flat plate CHT calculations. For all the analyzed cases the predicted wall temperatures were in agreement with Fluent predictions by less than 1% error.

Finally Hydra CHT solver has been validated against experimental data on the NASA C3X NGV. Considering the inherent experimental and numerical uncertainties a very good agreement has been achieved, comparable with other validation results available in the open literature on the same case (see Wlassow et al. (2010), Facchini et al. (2004) and York & Leylek (2003)).

Chapter 9

CHT analysis of a cooled nozzle guide vane

The present chapter shows the results of a CHT analysis, using the developed code, of a 2D cooled NGV. The geometry of the NGV is derived from the C3X blade used to validate the code through the insertion of a trailing edge cooling slot. The analysis will be focused on discussing the effect of CHT on the cooling design and flow characteristics.

The first section of the chapter will describe the trailing edge cooling design, followed by a discussion on the similarity conditions necessary to obtain engine representative results. In Sec. 9.3 the computational domain and boundary conditions used will be described followed by the grid independency study in Sec. 9.4. Finally the results of the simulations will be discussed in Sec. 9.5, followed by the final remarks.

9.1 Description of the trailing edge cooling

In his review Bunker (2013) highlighted the possible advantages obtainable through the adoption of *micro cooling* techniques as opposed to conventional *macro cooling* in terms of reduction of cooling mass flow and thermal stresses. The distinction between

macro and micro cooling is not clear-cut and is mainly related to the possibility to manufacture the cooling features through investment casting and the possibility that such passages will get obstructed by external particles. In very general terms the purpose of micro cooling is to substitute the traditional bulk internal cooling and discrete film cooling ejection with a distributed cooling system of smaller dimensions, and to bring the cooling flow closer to the external surface of the airfoil.

Despite the amount of patents available on micro-cooling designs, this kind of technology is far from an industrial use due to poor technological readiness, nonetheless, it is certainly appealing due to its possible advantages over conventional cooling designs as highlighted by Bunker (2013) and constitutes an interesting research path for future directions in advanced engine cooling technology (Xu et al. (2015)).

One of the most diffused micro cooling concepts is the *double-wall* or *near-wall* cooling (Kvasnak et al. (2003), Kingery et al. (2015), Ekkad et al. (1999), Mao et al. (2007), Ieronymidis et al. (2010)). In this kind of system cooling passages are moved in a thin gap between the external wall of the blade and the internal one. The internal cavities of the blade are then used only as plenums to supply air to the double-wall cooling system. An example of double-wall cooling design is presented in Fig. 9.1.

Due to the low coolant mass flow per cooling passage, the vicinity of the passage itself to the heat source and the distributed nature of such cooling design, it is of interest to understand the different performances of a double-wall cooling system under adiabatic and CHT conditions. For this reason the C3X NGV has been equipped with a double-wall trailing edge cooling system that has been purposely designed and is shown, with the relevant dimensions in Fig. 9.2. The figure shows also in red, a schematic of the coolant flow paths together with a legend for the used subscripts.

The trailing edge cooling system has two inlets that discharge to three film cooling holes on the pressure side and five inlets discharging on the trailing edge slot for the pressure side cooling. The relevant non-dimensional dimensions of the cooling system

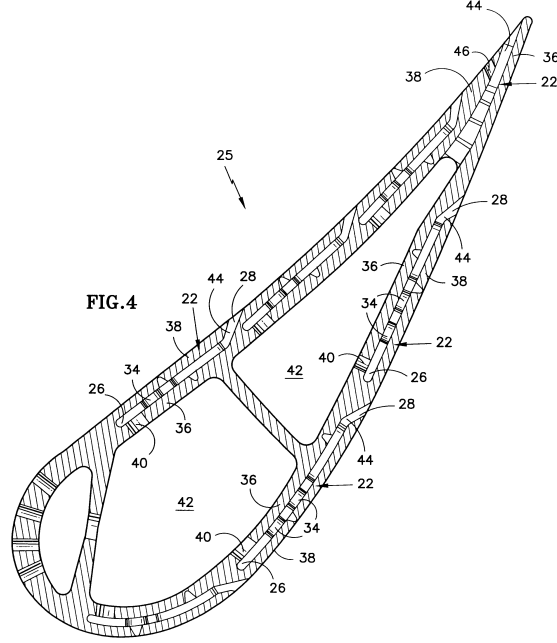


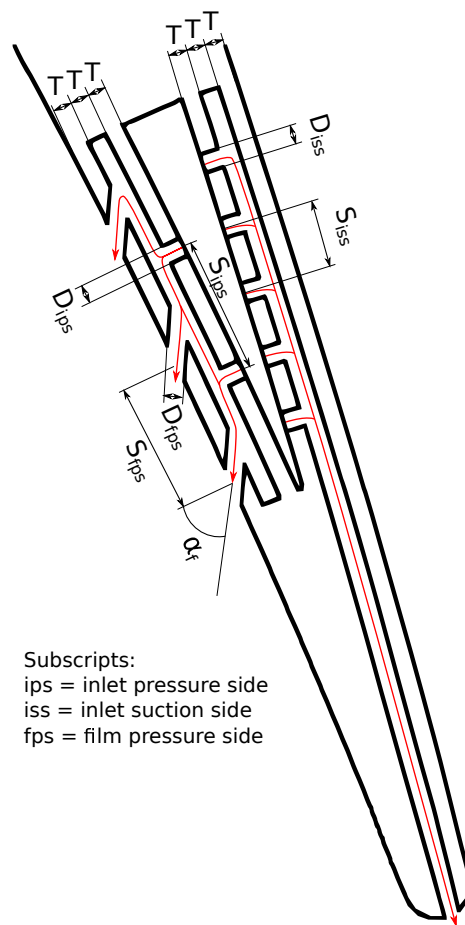
Figure 9.1: Example of double-wall cooling scheme from Kvasnak et al. (2003).

are summarized in Table 9.1.

The non-dimensional spacing between the cooling features has been chosen in line with the few papers found on double-wall cooling designs (see Land et al. (2010) and Mao et al. (2007)). The choice of using a value of the wall thickness of 1 *mm* comes from the fact that the original design of the C3X NGV has a chord of roughly 14 *cm* (see Hylton et al. (1983)), roughly twice of a realistic aero-engine NGV. It has thus been considered that the cooling design was scaled by 2:1 with respect to

D_{ips}/T	1	[-]
S_{ips}/D_{ips}	7	[-]
D_{fps}/T	1	[-]
S_{fps}/D_{fps}	7	[-]
α	60	[°]
D_{iss}/T	1	[-]
S_{iss}/D_{iss}	3.5	[-]
T	1	[mm]

Table 9.1: Summary of dimensions of the trailing edge cooling design. Symbols are explained in Fig. 9.2.



Subscripts:
 ips = inlet pressure side
 iss = inlet suction side
 fps = film pressure side

Figure 9.2: Scheme of the trailing edge cooling design with relevant dimensions and main coolant flow paths (red).

an hypothetical real engine-sized design with a dimension of 0.5 *mm* for the wall thickness. Thus the used 1*mm* in the scaled model. A wall thickness of 0.5 *mm* for the outer wall of the engine-sized NGV has been considered realistic as suggested also by Bunker et al. (2004) in his analysis of a double-wall cooling design.

9.2 Similarity conditions

The performed simulations have been run at low temperature conditions - as the ones performed for the validation case shown in Sec. 8.5.4 - to have realistic Reynolds number, Mach number and temperature ratio considering the scaled geometry.

For this reason the value of the thermal conductivity used in the solid domain had to be chosen carefully, to make sure that Bi , the Biot number for the simulated cases would match the one at engine realistic conditions. It is in fact well established the importance of matching Bi when CHT effects are of interest (see Albert et al. (2004) and Ramachandran & Shih (2015)).

Particularly, starting from a simple 1D thermal analysis of a typical cooling configuration, Dyson et al. (2014) show how the overall cooling effectiveness $\phi = (T_\infty - T_w)/(T_\infty - T_c)$ is dependent on both the Biot number and the ratio h_e/h_p between external and the plenum-side HTC's. The two issues of matching those two quantities will be presented in the following sections.

Matching of the external Biot number (Bi in what follows) will be illustrated in Sec. 9.2.1, followed by a discussion on the problems in matching as well the internal Biot number (Sec. 9.2.2). By internal Biot ($Bi_{internal}$ in what follows) the author refers to the Biot number based on the HTC in the double-wall internal cooling passages, from the coolant inlet to its exit. In Sec. 9.2.3 the issue of matching the ratio h_e/h_p will be presented. The reader should note that h_p , the HTC on the plenum walls, should not be confused with the internal HTC that is part of the definition of

$Bi_{internal}$.

9.2.1 External Biot number matching

The Biot number Bi represents the ratio of conductive thermal resistance in the solid and the thermal resistance of external convection process. It is defined as:

$$Bi = \frac{h_e L}{k_s} \quad (9.1)$$

where h_e is the external HTC and k_s is the thermal conductivity of the solid material, L is a conveniently chosen length scale, usually taken as the metal thickness as it is a representative length for the conduction process.

Matching of Bi from test to engine conditions requires that:

$$Bi_{test} = Bi_{engine} \rightarrow \frac{2 h_{etest}}{k_{stest}} = \frac{h_{engine}}{k_{sengine}} \quad (9.2)$$

where the geometrical scaling of the test model has been already considered.

The two convective heat transfer coefficients h_{etest} and h_{engine} can be expressed considering their respective Nusselt numbers. Equation (9.2) becomes then:

$$\frac{Nu_{test} k_{ftest}}{k_{stest}} = \frac{Nu_{engine} k_{fengine}}{k_{sengine}} \quad (9.3)$$

Matching the external flow Re , Pr and TR ensures matching of the flow Nu . This allows to express the solid conductivity at test conditions as:

$$k_{stest} = k_{ftest} \frac{k_{sengine}}{k_{fengine}} \quad (9.4)$$

Equation 9.4 shows how, as long as the Nu similarity is respected, the solid conductivity scales with the ratio of the air conductivity at engine and test conditions.

Considering $T_{0in} = 1600 [K]$ as engine representative, and a temperature of $800 [K]$ for test conditions, evaluating air conductivity at these temperatures using data from Kadoya et al. (1985) a value of:

$$k_{stest} \simeq 14.0 [W/m - K] \quad (9.5)$$

is obtained. This value for the thermal conductivity of the solid domain has been used throughout the analysis. The values of the solid density and specific heat have been kept identical to the values used for the validation cases (see Tab. 8.5) as in any case they do not influence the steady state thermal field.

9.2.2 Internal Biot number matching

It is of interest to discuss also whether the used solid thermal conductivity allows to match as well the internal Biot number.

For $Bi_{internal}$ the same reasoning applied in the previous section can be used, leading to the following relation:

$$Bi_{internal} = \frac{Nu_{internal} k_c}{k_s} \quad (9.6)$$

where the subscript c refers to coolant conditions now, rather than to external flow and the subscript s refers to the solid.

From Eqn. 9.6 a first problem regarding the difficulty of matching the internal Bi on a complex cooling design is clear: the internal Nusselt number depends on the layout of the cooling system. In a case like the present, where the suction and pressure side cooling systems have different features, a single $Nu_{internal}$ may be hard to define, as such exact matching of the internal Bi on both cooling systems becomes impractical.

Assuming that a single functional relation of the kind of Eqn. (9.7) can be sought

for an average $Nu_{internal}$ the problem can be simplified. Under this assumption, matching the internal Nu requires matching the corresponding coolant Reynolds number. This considers that the Prandtl number Pr is largely invariant between test and engine conditions (Albert et al. (2004)) and the temperature ratio TR is identical in the two cases.

$$Nu_{internal} = f(Re_{internal}, Pr_{internal}, TR_{internal}) \quad (9.7)$$

The internal Reynolds number can be defined as:

$$Re_{internal} = \frac{(\rho U)_c D}{\mu_c} = \frac{BR (\rho U)_{external} D}{\mu_c} \quad (9.8)$$

considering that the mass flow in the cooling channel $(\rho U)_c$ is related to the vane mass flow through the blowing ratio BR . Finally, from the external Reynolds number definition $Re_{external} = ((\rho U)_{external} L) / \mu_{external}$, the internal Re can be expressed as:

$$Re_{internal} = \frac{BR Re_{external} \mu_{external} D}{\mu_c L} \quad (9.9)$$

Given that the external Re is matched between engine and test conditions, as is the ratio of relevant dimensions, matching of the internal Bi depends on matching the following quantities:

- BR
- $\mu_{external} / \mu_c$
- k_c / k_s

It appears, and this is confirmed in the results of Sec. 9.5.1, that the blowing ratios are closely coupled with internal heat transfer and do depend on (and in turn influence) the internal Bi . However, as shown in Sec. 9.5.1, the dependence of blowing ratios

appears to be mild for the present case and they can be considered to be matched within engineering approximation as long as the coolant pressure ratios are identical between test and engine conditions.

For what concerns the matching of the viscosity ($\mu_{external}/\mu_c$) and conductivity (k_c/k_s) ratios, when high flow to coolant temperature differences are of interest, both the viscosity and conductivity variations with temperature depart from a linear behavior. As such both the ratios $\mu_{external}/\mu_c$ and k_c/k_s cannot be identical between experiments and test conditions if the temperature ratio has to be matched as well.

Given the practical difficulty to match the internal Biot number, as explained, it is worth trying to quantify, in a very approximate way, the extent and possible implications of this mismatch for the present case. A precise quantification is in fact not simple as the effect of a different viscosity ratio on $Bi_{internal}$ depends on the functional dependence of $Re_{internal}$ on Nu (Eqn. (9.7)). This function has been assumed at the beginning of this analysis to be unique but in fact, given the present cooling design, the pressure side and suction side cooling cannot have the same kind of dependence of Nu on the relevant non-dimensional quantities. In theory there are two distinct internal Biot numbers that should be matched: one for the pressure side cooling and one for the suction side. This said, based on the available correlations and engineering considerations, an approximate estimate of the $Bi_{internal}$ mismatch on the more critical of the two cooling systems can be obtained.

Given the shielding provided by the film, it is expected the pressure side cooling system to be less influenced by CHT effects than the suction side slot. This is confirmed by the results discussed in Sec. 9.5. As such, a mismatch in the internal Bi on the pressure side is less influential than a mismatch on the suction side cooling. For this reason the following discussion will be aimed at giving an approximate value for the Bi mismatch on the suction side slot.

Average Nusselt numbers in internal cooling passages with impingement are nor-

mally correlated to a power of the jet Reynolds number: $Nu_{internal} = f(Re_{internal}^n, \dots)$ (e.g. Florschuetz et al. (1981)), where the value of the exponent n depends on the particular correlation/cooling configuration.

From the previous discussion it is possible to express the ratio between the internal Biot number at test and experimental conditions as:

$$\frac{(Bi_{Internal})_T}{(Bi_{Internal})_E} \simeq \left[\frac{(\frac{\mu_{external}}{\mu_c})_T}{(\frac{\mu_{external}}{\mu_c})_E} \right]^n \frac{(\frac{k_c}{k_s})_T}{(\frac{k_c}{k_s})_E} \quad (9.10)$$

Given the length of the trailing edge slot, and the relative low impingement effect due to high cross flow from the most upstream suction side coolant inlet (*ISS1* in Fig. 9.3), an average Nu for the suction side cooling can be calculated considering it as a simple straight cooling channel. Typically used for turbulent flow in straight channels is the Dittus-Boelter correlation (Dittus & Boelter (1985)), correlating the Nu with $Re^{0.8}$.

Considering the values of the solid thermal conductivity at engine and test conditions as from Sec. 9.2.1, evaluating air properties at the respective relevant temperatures and using $n = 0.8$, Eqn. (9.10) can be evaluated as:

$$\frac{(Bi_{Internal})_T}{(Bi_{Internal})_E} \simeq 1.3 \quad (9.11)$$

Due to the impossibility of correctly matching both the internal and the external Biot numbers, the internal Bi results in being approximately 30% higher in the test conditions than at engine realistic temperatures. This, in general terms, means an increased ratio between the conductive and the convective thermal resistance of the coolant flow, decreasing the internal cooling heat pickup with respect to engine conditions.

Quantification of the effect of such mismatch (and confirmation of its magnitude) of the internal Biot number on the predicted overall cooling effectiveness is outside

the scope of this chapter. Drawing further conclusions on the implications of the impossibility to match $Bi_{internal}$ is not possible also as, to the author's best knowledge, no relevant papers on the topic are available in the open literature.

9.2.3 External to internal HTC's ratio matching

In the performed computations the plenum is a boundary for the domain and HTC's on its walls can be specified as boundary conditions. Differently from the double-wall internal cooling passages, where the wall Nu is linked to the flow solution, the possibility to impose h as a boundary condition allows to "decouple" the Nusselt number from the Reynolds number. As such one can think of the specified wall HTC as resulting from a flow in the plenum that has matched Re with engine realistic conditions.

If the Reynolds numbers are matched, in order to match the ratio h_e/h_p between the external HTC and the HTC at the plenum walls, it is sufficient to match the ratio k_{f_e}/k_{f_i} between the thermal conductivity of external flow and of the coolant (see Albert & Bogard (2013)).

Considering the case $T_{0in} = 1600 [K]$ and $T_c = 720 [K]$ as engine realistic and $T_{0in} = 800 [K]$ and $T_c = 360 [K]$ ¹ a value of:

$$\frac{h_e}{h_p} = \frac{\left(\frac{k_{f_e}}{k_{f_i}}\right)_{engine}}{\left(\frac{k_{f_e}}{k_{f_i}}\right)_{test}} \simeq 1.1 \quad (9.12)$$

is found.

This mismatch is only due to the non-linear dependence of air conductivity with temperature and, as such, is common to all experimental and numerical analyses involving low temperature scaling. Using the same approach described in Albert & Bogard (2013), considering the value of the external Biot number for the present

¹This value is an average value for the coolant temperature on all the internal cooling holes.

case, it is possible to estimate the maximum difference in cooling effectiveness $\Delta\phi = \phi_{engine} - \phi_{test}$ deriving from this mismatch as being $\Delta\phi \simeq 0.01$. Such $\Delta\phi$ translates in a $< 1\text{ K}$ wall temperature difference when scaling the overall effectiveness value to obtain engine-realistic wall temperature.

9.3 Computational domain and boundary conditions

The computational domain and type of boundary conditions used for the external flow are identical to the one used in the C3X validation (see Fig. 8.2). All solid walls have been modeled as viscous walls. All walls standing at a fluid-solid interface have been considered as conjugate interfaces except for adiabatic simulations.

Differences are related to the cooling system of the blade. In Fig. 9.3 a legend of the internal cooling walls and inlets is shown. All cooling holes (1-7 in Fig. 9.3) are modeled as convective walls, as well as the trailing edge plenum (P in Fig. 9.3). The feeding inlets for the pressure side film cooling (ips1 and ips2 in Fig. 9.3) and for the trailing edge slot (iss1 to iss5 in Fig. 9.3) are modeled as total pressure inlets. The total pressure in each of these holes has been tuned to:

- obtain blowing ratios as even as possible for the three film cooling holes,
- allow an even discharge by all five inlets on the trailing edge slot.

Two flow conditions have been analyzed corresponding to two different outlet isentropic Mach numbers:

- $M_{is_{out}} = 0.9$ low Mach number case (denoted as *LM* in the remaining of the chapter),
- $M_{is_{out}} = 1.1$ high Mach number case (denoted as *HM* in the remaining of the chapter).

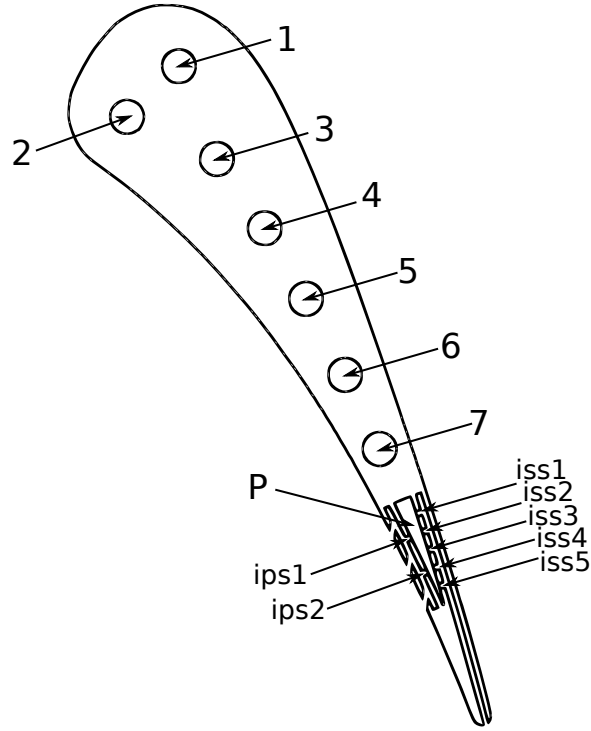


Figure 9.3: Legend for the boundaries of the computational domain.

The external flow conditions of the *LM* case correspond to the same conditions of the validation case presented in Sec. 8.5.4 while the conditions for the *HM* case have been taken from experimental case No. 5411 in Hylton et al. (1983).

For each of these flow conditions the pressure in the cooling inlets has been varied to obtain two sets of Blowing Ratios (*BR*) in the film cooling and trailing edge slots. The two blowing ratio cases will be referred to as:

- *LBR* for the low BR case
- *HBR* for the high BR case

In the *LBR* cases a coolant to mainstream mass flow ratio of about 3% (1% for the impingement, 2% for the film cooling) was achieved. The *HBR* cases brought the coolant mass flow ratio to about 6% (2% for the impingement, 4% for the film cooling). This was roughly independent of the external flow conditions. The mass flow ratios achieved by the present design were quite high given the scope of cooling

	<i>LM</i>	<i>HM</i>
M_{out}	0.9 [–]	1.1 [–]
Re_{in}	0.4×10^6 [–]	0.5×10^6 [–]
Tu_{in}	6.5% [–]	6.5% [–]
TR	0.7 [–]	0.7 [–]

Table 9.2: Flow conditions for the *LM* and *HM* cases.

	<i>LM</i>	<i>HM</i>
p_{0in}	2.45×10^5 [Pa]	3.11×10^5 [Pa]
T_{0in}	802.0 [K]	798.0 [K]
μ_t/μ	1.0×10^3 [–]	1.0×10^3 [–]
p_{out}	1.43×10^5 [Pa]	1.55×10^5 [Pa]

Table 9.3: Boundary conditions imposed at inlet and outlet for the *LM* and *HM* cases.

only the trailing edge region, but it has not been possible to obtain lower values without having hot gas ingestion in the film holes.

The conjugate simulations will be labeled as *CHT*, the adiabatic ones as *AD*.

Flow conditions for the *LM* and *HM* cases are summarized in Table 9.2. The relative inlet/outlet boundary conditions are shown in Table 9.3. The boundary conditions imposed at the pressure inlet and outlet for are summarized in Table 9.3.

In Table. 9.4 the total pressures at the trailing edge cooling inlets are expressed as pressure ratios PR defined as: $PR = p_{0in_{cooling}}/p_{0in}$ for the two blowing ratio cases.

For all the analyzed *CHT* cases the convective boundary conditions imposed on the cooling holes from 1 to 7 in Fig. 9.3 are identical to the convective boundary conditions assigned on the same holes for the C3X validation case (see Table. 8.4). The values of h and T_f imposed on the plenum walls (P in Fig. 9.3) are discussed in the following.

As the trailing edge cooling geometry has been created substituting the last three cooling holes of the original C3X case, the value of T_f for the plenum has been taken as the mass-weighted average of the coolant temperatures for holes 8 to 10 in the

	<i>LBR</i>	<i>HBR</i>
PR_{iss1}	0.7 [–]	0.9 [–]
PR_{iss2}	0.7 [–]	0.85 [–]
PR_{iss3}	0.7 [–]	0.8 [–]
PR_{iss4}	0.7 [–]	0.8 [–]
PR_{iss5}	0.675 [–]	0.75 [–]
PR_{ips1}	0.9 [–]	1.05 [–]
PR_{ips2}	0.85 [–]	1.0 [–]

Table 9.4: Pressure ratios at inlet of trailing edge cooling inlets for the *LBR* and *HBR* cases.

original C3X case, corresponding to $T_f = 411.7 [K]$.

The coolant HTC on the plenum has been determined using the correlation proposed by Humble et al. (1951) to calculate Nu in smooth pipes (shown in Eqn. 9.13).

$$Nu = 0.034(Re)^{0.8}(Pr)^{0.4}\left(\frac{T_f}{T_w}\right)^{0.55} \quad (9.13)$$

The Reynolds number Re in Eqn. 9.13 is based on the hydraulic diameter of the pipe and considers the bulk coolant velocity (based on the bulk temperature T_f) but uses the density and viscosity calculated at the wall temperature. For this case, considering the shielding provided by the film cooling it has been assumed that in all cases $T_w = T_f$. The values of Nu have been obtained assuming a value of the coolant Mach number in the plenum $M = 0.1$. Given an average total pressure in the plenum (taken as an average of the pressure ratios shown in Table. 9.4) the obtained h values are shown in Table 9.5. It should be noted that, despite the same pressure ratios for coolant flow are used between the *LM* and *HM* cases, these have different inlet total pressures, and this results in four different values for HTC at the plenum walls. It is worth highlighting that this should not be considered an inconsistency when comparing the results as indeed a different wall HTC in the plenum would be expected if a full conjugate analysis (including also flow simulation of the internal cooling passage) was to be carried out.

	<i>LBR</i>	<i>HBR</i>
<i>LM</i>	388.9 [W/m ²]	495.7 [W/m ²]
<i>HM</i>	471.9 [W/m ²]	601.4 [W/m ²]

Table 9.5: Values of HTC imposed at the plenum wall for all analyzed CHT cases.

9.4 Mesh dependency study

As discussed in the previous chapter, the developed CHT solver can only handle 3D domains. For this reason the 2D geometry examined has been extruded in the normal direction and meshed using symmetry boundary conditions on the surfaces normal to the extrusion direction. The node counts that will be reported hereafter refer to the number of nodes of a single 2D section, not to the nodes on the whole 3D mesh.

Two grid densities have been considered: the first one having 400k and the finer one having 700k nodes. The mesh refinement has been mainly concentrated on the trailing edge region (and so the discussion about the mesh dependency tests) as the C3X runs performed on the case used for the solver validation allowed to establish a baseline mesh density for the present case. A comparison of the two grids used is shown in Fig. 9.4.

As the magnitude of gradients and the general flow features in the internal cooling region and in the solid are not expected to undergo drastic changes between all the analyzed cases, the mesh requirements for all of them would be to all practical extents identical in those regions. The choice of the test case for the mesh dependency tests fell on the *HM-HBR-CHT* case as the high Mach flow condition set the highest requirement in terms of external flow grid due to the presence of shocks, absent in the *LM* case.

In Fig. 9.5 the Mach number contours on the two grids analyzed are shown. Figure 9.6 shows the temperature contours in the trailing edge region obtained using the fine and coarse grids. In Fig. 9.7 the wall temperature distribution obtained on the

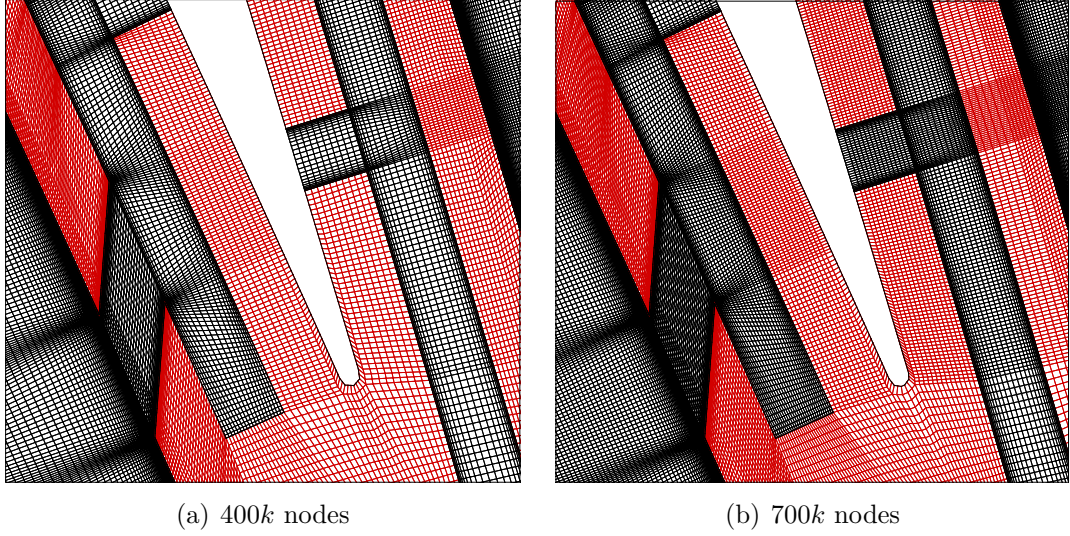


Figure 9.4: Details of the coarse and fine grids used for mesh independency tests.

coarse and fine grids are shown. All the results shown confirm the grid independency of the analyzed case.

9.5 Results

The simulations performed will be analyzed aiming to discuss the differences in the cooling performance and external aerodynamic flow characteristics between adiabatic and CHT solution.

In Sec. 9.5.1 the differences in the coolant flow main discharge characteristics will be analyzed. Section 9.5.2 will present a discussion of the film development on the pressure side through traversing at different distances from the ejection points as well as a traversing of the boundary layer on the suction side. Section 9.5.3 will present results regarding the wall shear on the previously analyzed traversing locations.

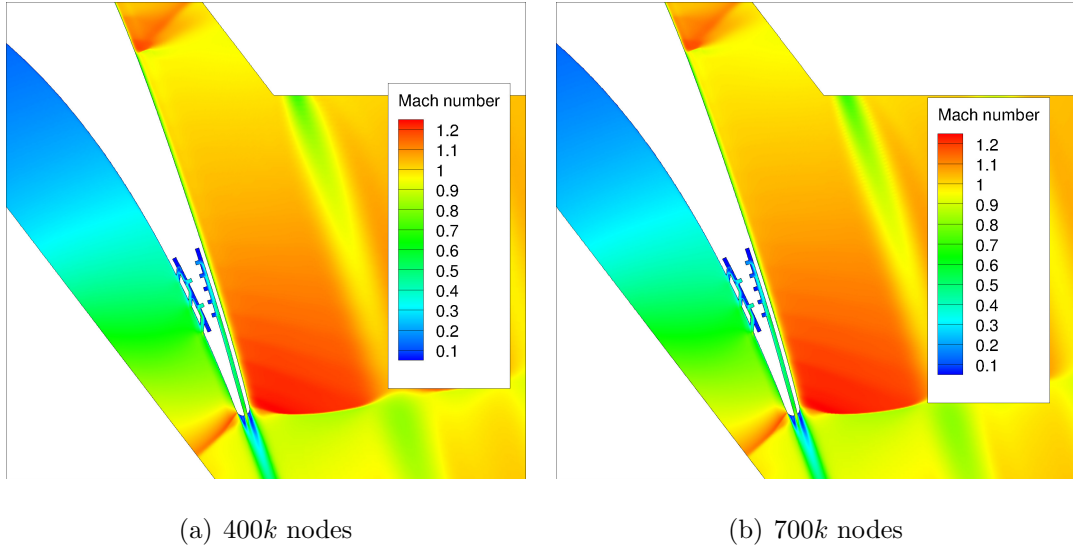


Figure 9.5: Mach number contours for the *HM-HBR-CHT* case on the coarse and fine grids.

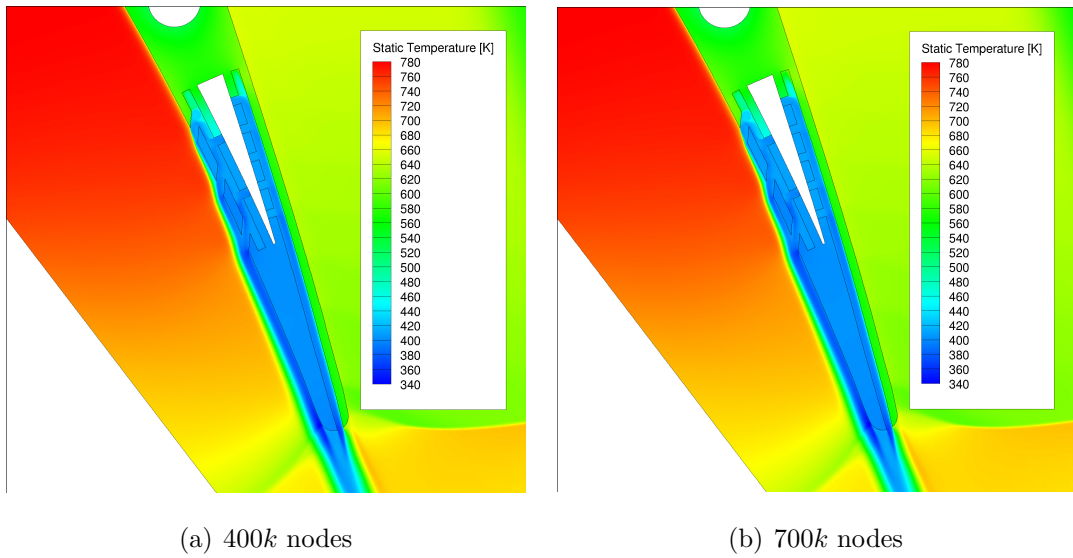


Figure 9.6: Temperature contours for the *HM-HBR-CHT* case on the coarse and fine grids.

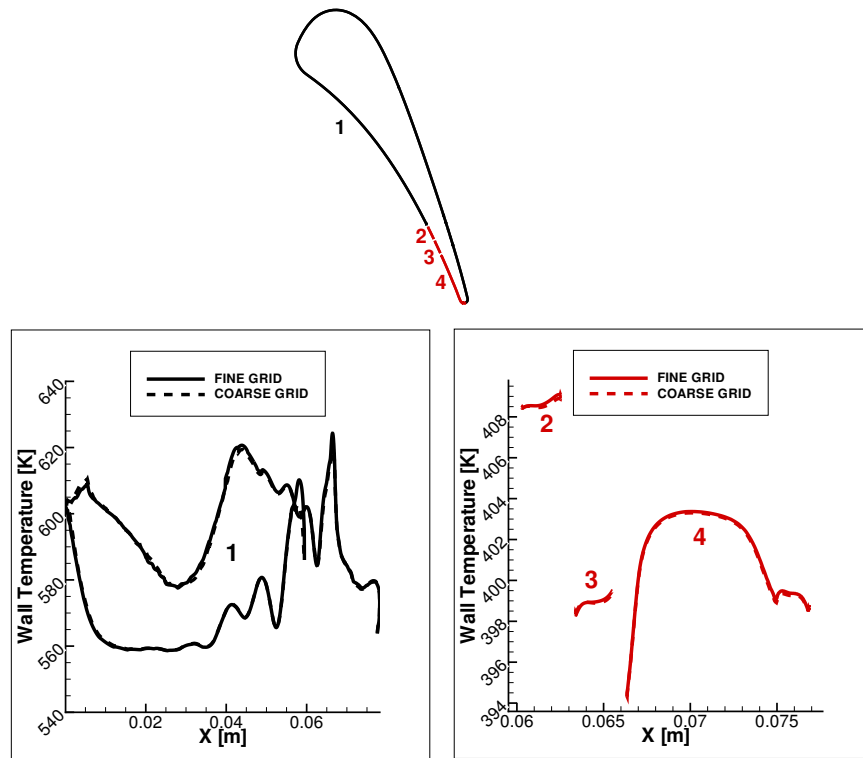


Figure 9.7: Wall temperature distribution for the *HM-HBR-CHT* case on the coarse and fine grids. Temperature on walls 2, 3 and 4 is shown in a second graph for clarity as the temperature on these walls is markedly lower than the rest of the blade.

9.5.1 Coolant discharge characteristics

In this section the differences in relevant coolant non-dimensional quantities at the coolant ejection holes between adiabatic and conjugate will be analyzed. Figure 9.8 shows the nomenclature adopted in the following to identify the discharge holes. The quantities analyzed are:

- Blowing ratio:

$$BR = \frac{(\rho U)_{CD}}{(\rho U)_{Out}} \quad (9.14)$$

- Momentum flux ratio:

$$I = \frac{(\rho U^2)_{CD}}{(\rho U^2)_{Out}} \quad (9.15)$$

- Density ratio:

$$DR = \frac{\rho_{CD}}{\rho_{Out}} \quad (9.16)$$

- Velocity ratio:

$$VR = \frac{U_{CD}}{U_{Out}} \quad (9.17)$$

where CD refers to the coolant discharge cut plane ($F1$, $F2$, $F3$ and S in Fig. 9.8) and Out to the pressure outlet of the computational domain. The choice of using the outlet rather than the inlet to non-dimensionalize the quantities has been done as outlet values are more representative of the local conditions at the coolant discharge planes. The quantities appearing in Eqs.(9.14) to (9.17) are mass-averaged values over the respective cut sections.

Table 9.6 presents a summary of all the discussed non-dimensional parameters for all the adiabatic simulations performed (both for $HM-LM$ and $LBR-HBR$ conditions). The percentage differences that will be discussed in the following are referred

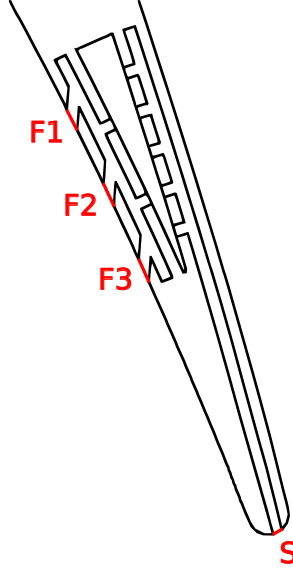


Figure 9.8: Nomenclature used for the coolant ejection holes.

to these values and calculated as:

$$Perc. \, diff. = \frac{AD - CHT}{AD} \times 100 \quad (9.18)$$

Figures 9.9 and 9.10 show a summary of the percentage differences in DR , VR , BR and I between adiabatic and CHT cases for LBR (blue) and HBR (red). Figure 9.9 refers to the high subsonic external flow (LM) case, while Fig. 9.10 refers to the transonic external flow (HM) conditions.

The percentage differences for all examined quantities show the similar trend of decreasing between the most upstream film cooling hole $F1$ to the last $F3$. This is to be expected as the foremost cooling hole is exposed to the highest heat load from external flow, while the others are shielded by the first one. This is particularly true for the HBR conditions, where the higher blowing ratio of the first film cooling hole (still not lifted from the wall surface as seen in Fig. 9.6) allows for an increased thermal protection to the downstream part of the blade.

(a) Film cooling discharge 1: $F1$				
	LM		HM	
	LBR	HBR	LBR	HBR
BR_{F1}	0.29	0.69	0.30	0.71
I_{F1}	0.04	0.20	0.04	0.19
DR_{F1}	2.19	2.48	2.48	2.77
VR_{F1}	0.13	0.27	0.12	0.26
(b) Film cooling discharge 2: $F2$				
	LM		HM	
	LBR	HBR	LBR	HBR
BR_{F2}	0.35	0.91	0.40	0.93
I_{F2}	0.07	0.35	0.08	0.32
DR_{F2}	1.97	2.39	2.28	2.67
VR_{F2}	0.17	0.38	0.16	0.35
(c) Film cooling discharge 3: $F3$				
	LM		HM	
	LBR	HBR	LBR	HBR
BR_{F3}	0.57	0.81	0.59	0.83
I_{F3}	0.17	0.33	0.16	0.30
DR_{F3}	2.24	2.24	2.48	2.49
VR_{F3}	0.26	0.37	0.23	0.33
(d) Trailing edge slot discharge: S				
	LM		HM	
	LBR	HBR	LBR	HBR
BR_S	0.52	0.91	0.76	1.04
I_S	0.16	0.46	0.35	0.62
DR_S	1.77	1.83	1.65	1.73
VR_S	0.30	0.50	0.46	0.60

Table 9.6: Values for the coolant parameters at discharge sections for the adiabatic simulations.

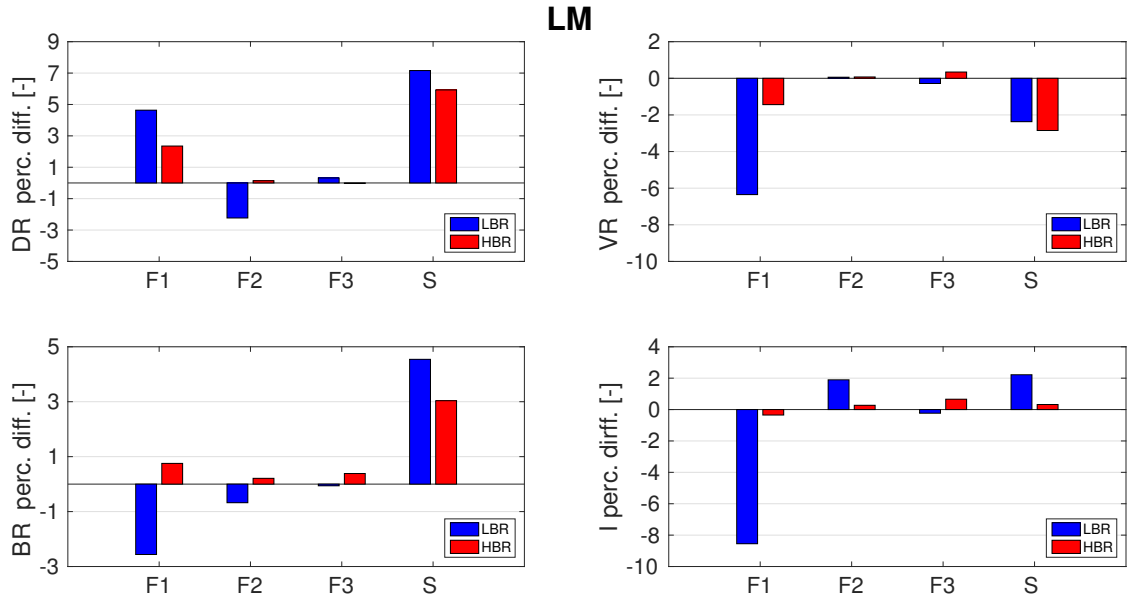


Figure 9.9: Comparison of CHT Vs Adiabatic percentage difference (Eqn (9.18)) in DR , VR , BR and I for all coolant discharge sections ($F1$, $F2$, $F3$, S). Lower outlet Mach number case (LM).

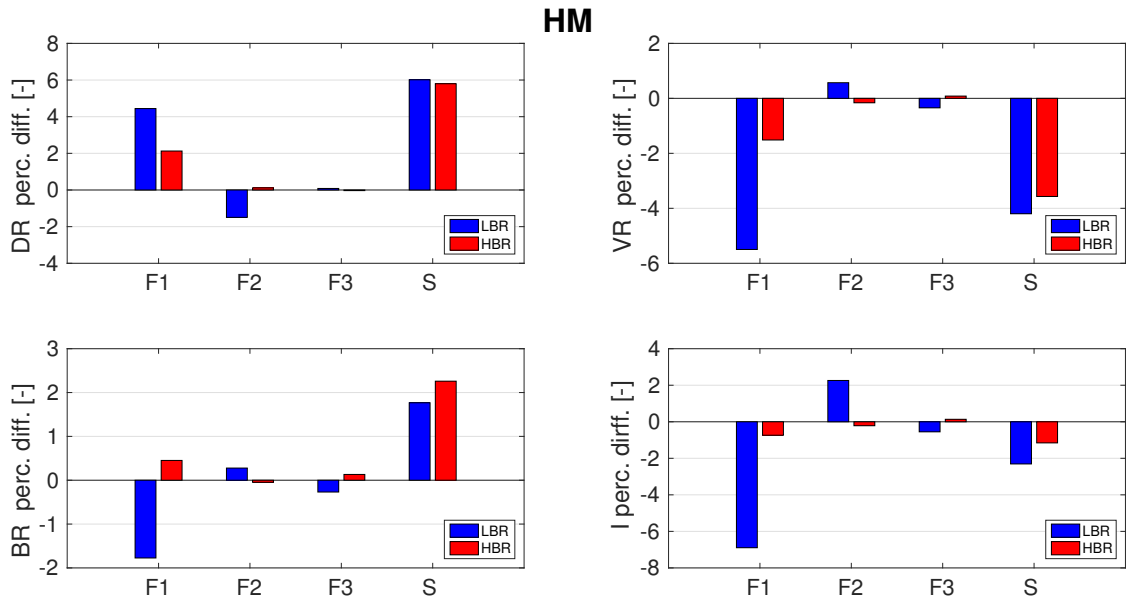


Figure 9.10: Comparison of CHT Vs Adiabatic percentage difference (Eqn (9.18)) in DR , VR , BR and I for all coolant discharge sections ($F1$, $F2$, $F3$, S). Higher outlet Mach number case (HM).

From the comparison of Fig. 9.9 and Fig. 9.10 it can be seen that similar conclusions can be drawn between the *LM* and *HM* cases. Marginally lower percentage differences can be observed in the *HM* cases with respect to the *LM* and this is consistent with the slightly higher coolant mass flow ejected at film cooling locations for the *HM* cases (see *BR* values in Table 9.6 as a reference).

The density ratio shows a positive percentage difference at *F1* for all cases. This is consistent with the lower density expected at the cooling hole exit for the conjugate case with respect to the adiabatic one. A maximum of about 5% *DR* difference is observed for the *LM* – *LBR* case. Interestingly the *LBR* cases show a negative *DR* difference, meaning that the average density at the cooling hole outlet is higher for the *CHT* case than the adiabatic one. The reason for this result at low blowing ratio is explained considering the following.

Figure 9.11 shows that, contrarily to what happens at *HBR* conditions, a mild ingestion of hot gas happens in the second film cooling hole *F2*, with some of the hot gas from upstream passing the exit plane of the cooling hole. The reason of the higher average density at *F2* seen in the adiabatic case is due to the fact that, despite for the *CHT* case the coolant ejected is warmer, the hot gas ingested has a lower temperature than the adiabatic case, with the second effect prevailing on the first one on the average density at *F2*.

The reason for the higher temperature of the ingested hot gas in the adiabatic case is explained considering the film effectiveness of the first film cooling hole. At low blowing ratios the first film works at around $BR = 0.3$ and a momentum flux ratio $I = 0.04$ for all cases. An analysis of the film effectiveness of the film (Fig. 9.12) reveals that the film coverage drops quite rapidly downstream of the wall. As the adiabatic effectiveness decreases the temperature of the wall gas approaches the one of the external flow in the adiabatic case. For the *CHT* case, however, even if the coverage of the film is mild, the effect of impingement behind the blade wall lowers the

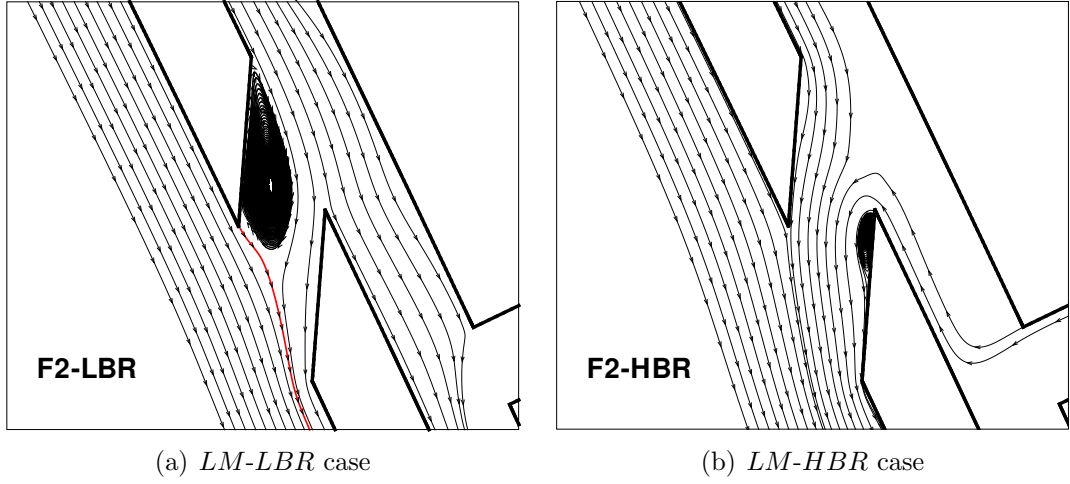


Figure 9.11: Streamlines at the second film cooling ejection hole $F2$ for the LBR and HBR cases. In red for the $LM-LBR$ case is the limiting streamline between external hot gas and coolant flow.

temperature of the blade and of the external flow to values lower than the adiabatic case. For this reason the ingested flow revealed to be cooler in the CHT case than in the adiabatic. This effect is not seen for the HBR case as the upstream film coverage is extended due to the increased BR and also there is no ingestion in $F2$ as shown in Fig. 9.11(b).

Smaller differences are observed for the $F3$ discharge section in all analyzed cases due to the protection offered by the upstream film.

The trailing edge cooling slot S section features the highest variations of density ratio, about 7% for the $LM - LBR$ case. This is expected due to the length of the internal cooling slot, increasing the heat pickup of the coolant.

As a result of the decreased DR , the coolant exit velocity is increased at both $F1$ and S for the CHT cases. Both the LM and HM flow conditions see a difference of about -6% at $F1$.

The highest difference in VR at the trailing edge discharge is observed for the $HM - LBR$ case, showing a -4% difference between adiabatic and CHT conditions

The differences in density ratios and velocity ratios contribute to the different

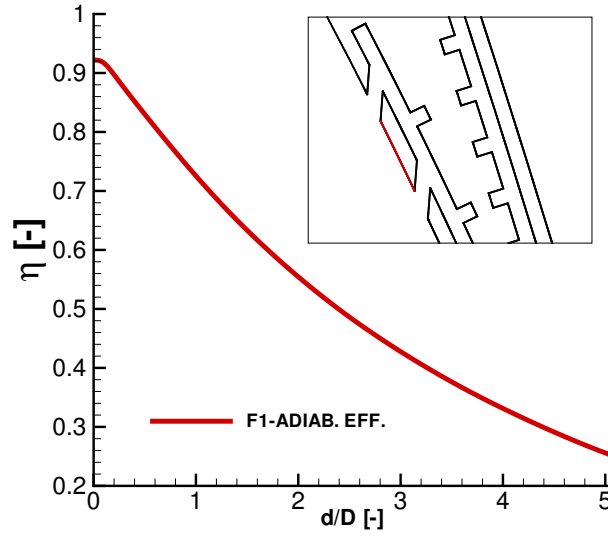


Figure 9.12: Adiabatic film effectiveness for the first film ejection, *LM-HBR* case.

blowing ratios (BR) and momentum flux ratios (I). Particularly the *LM – LBR* case features a 3% lower blowing ratio in the adiabatic case than in the CHT case and a 8% lower momentum flux ratio.

9.5.2 Boundary layer traversing

Traversing of the film cooling flow and the boundary layer at the suction side has been done to see the impact of CHT on the velocity and ρU distributions on the boundary layer flow. Importance has been given to ρU cuts as displacement thickness has a relative higher importance on losses than momentum thickness for compressible flows as highlighted by Denton & Xu (1990).

Traversing of the film cooling flow has been made at four different sections to evaluate the effect of CHT on the film development. The position and the extension in the normal direction of the traversing cuts is shown in Fig. 9.13 together with the suction side traversing section *SS*. The cuts have been made on planes normal to the local wall direction.

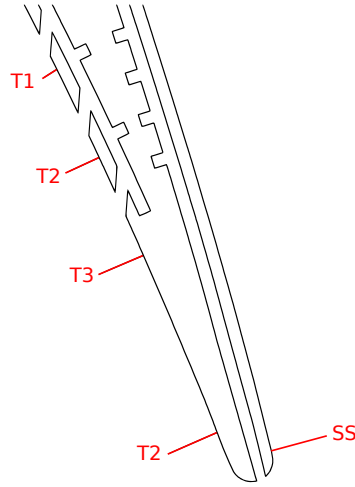


Figure 9.13: Positions and extension of film traversing cuts.

The results of the film cooling and suction side traversing are discussed separately in the next sections.

9.5.2.1 Film cooling traversing

Figures 9.14 and 9.15 show the distributions of U in all the four film traversing positions for both HM and LM external flow conditions with low film blowing ratio (LBR) and high blowing ratio (HBR) respectively.

The low blowing ratio cases show, specially at $T1$ and $T2$, almost no trace of the film jet, as a result of the low momentum flux ratio of the ejected film at these conditions. The film jets are more visible in Fig. 9.15, that refers to high film blowing ratios. Generally very little differences can be observed in the velocity distributions between adiabatic and CHT conditions for all the cases. Interestingly the higher differences are seen at $T1$ in the upper part of the boundary layer. Those differences, observed at all conditions are to be ascribed to the different pressure side boundary layer profile developed upstream of the first cooling hole.

Figures 9.14 and 9.15 show the distributions of ρU in all the four traversing positions for both HM and LM external flow conditions with low film blowing ratio

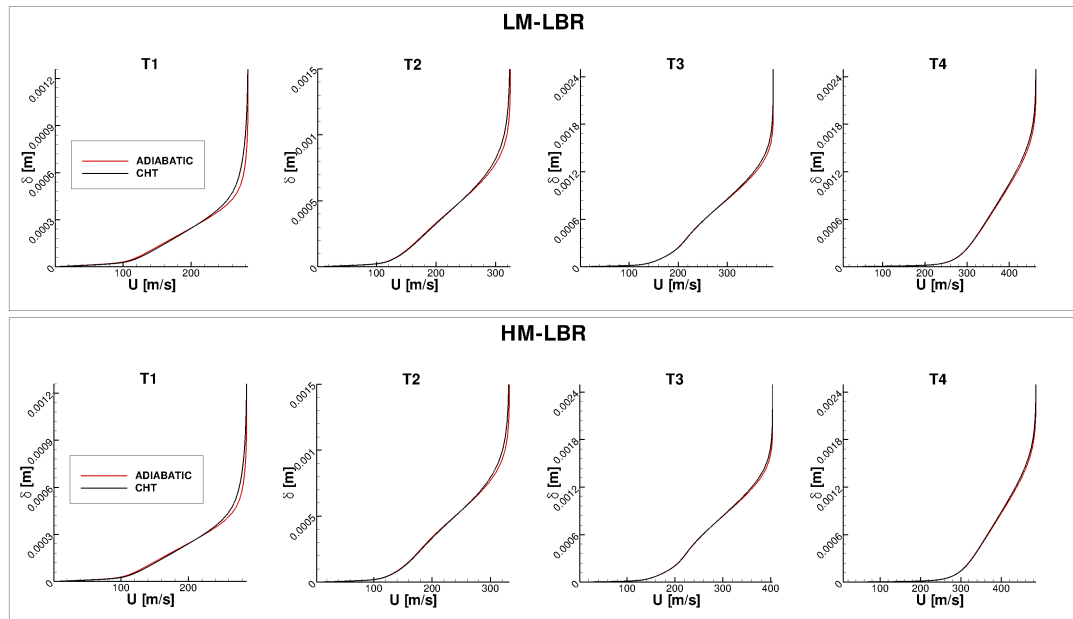


Figure 9.14: Distributions of U through the four traversing cuts $T1$, $T2$, $T3$, $T4$ for the $LM-LBR$ and $HM-LBR$ cases.

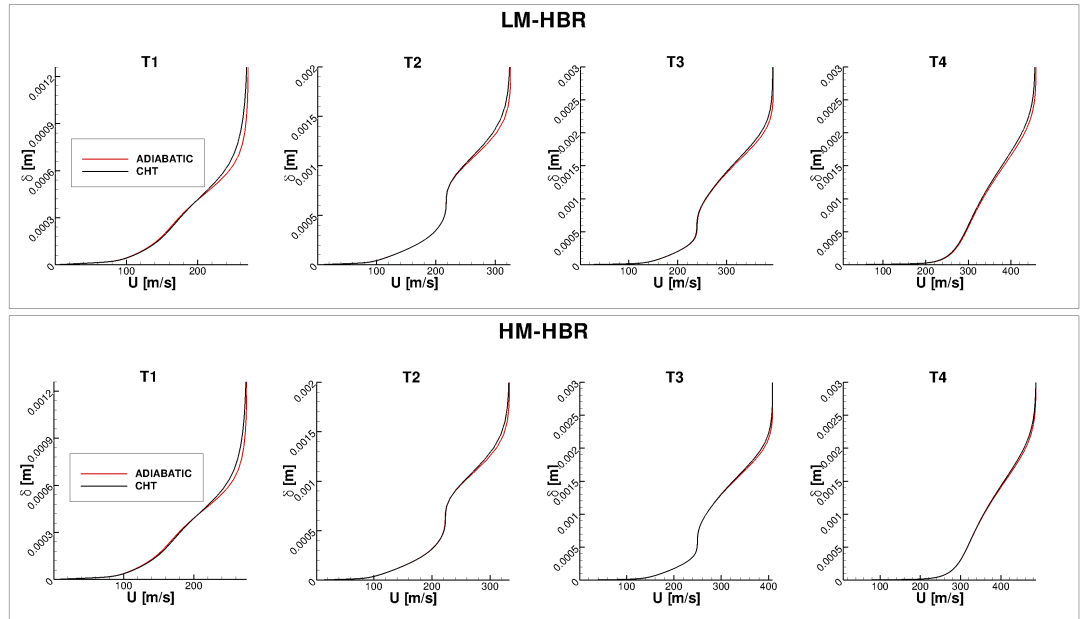


Figure 9.15: Distributions of U through the four traversing cuts $T1$, $T2$, $T3$, $T4$ for the $LM - HBR$ and $HM - HBR$ cases.

(*LBR*) and high blowing ratio (*HBR*) respectively. Each plots indicates also the relative difference in the boundary layer mass flow rate between the adiabatic case and the conjugate one.

As expected, due to the density changes in the boundary layer, the ρU distributions show much more differences than the velocity field alone. The differences are prominent at the first cut *T1*, and tend to shrink as the film develops. This is a consequence of the fact that the shielding provided by the upstream film cooling rows, in this configuration, is such that downstream of the first ejection, virtually no differences can be observed between a CHT and an adiabatic case.

The differences in the ρU profiles show that, where CHT effects are relevant, the CHT boundary layer has always a thicker mass flux profile, with differences close to 3% at *T1*. This corresponds to a 3% lower displacement thickness that certainly has a positive impact on the local profile losses, since they depend, on transonic blades, linearly on the displacement thickness as described by Mee et al. (1992). In this view the effect of CHT on profile losses increases as the portion of the profile interested by different density boundary layer increases.

9.5.2.2 Suction side boundary layer traversing

Figure 9.18 shows the velocity and ρU profiles in the late suction side boundary layer cut *SS* for both *LM* and *HM* flow conditions under low and high blowing ratios.

While for the *LM* case the differences in the velocity field are limited, the high Mach number case shows fuller velocity profiles for the *CHT* cases with respect to the adiabatic. It has to be noted that the *SS* cut has been done on the very late suction side, very close to the trailing edge that in the *HM* case is subject to recompression shocks. The position of the suction side traversing cut with respect to the trailing edge shock is illustrated in Fig. 9.19(a), while in Fig. 9.19(b) the superimposed Mach number flood lines for the adiabatic and CHT simulation in the *HM-HBR* case are

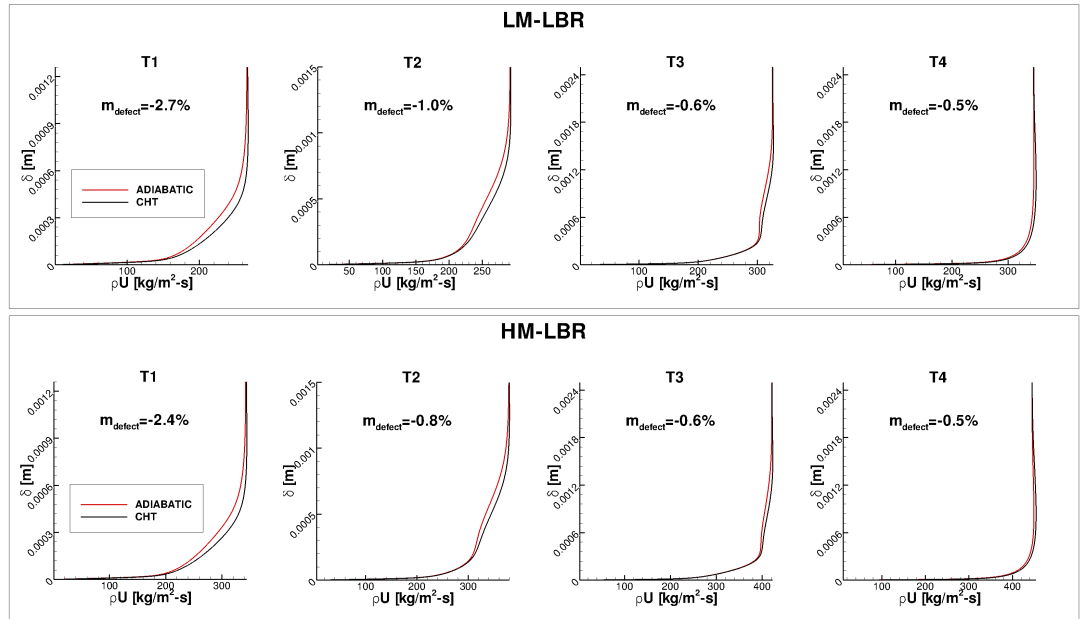


Figure 9.16: Distributions of ρU through the four traversing cuts $T1$, $T2$, $T3$, $T4$ for the $LM - LBR$ and $HM - LBR$ cases.

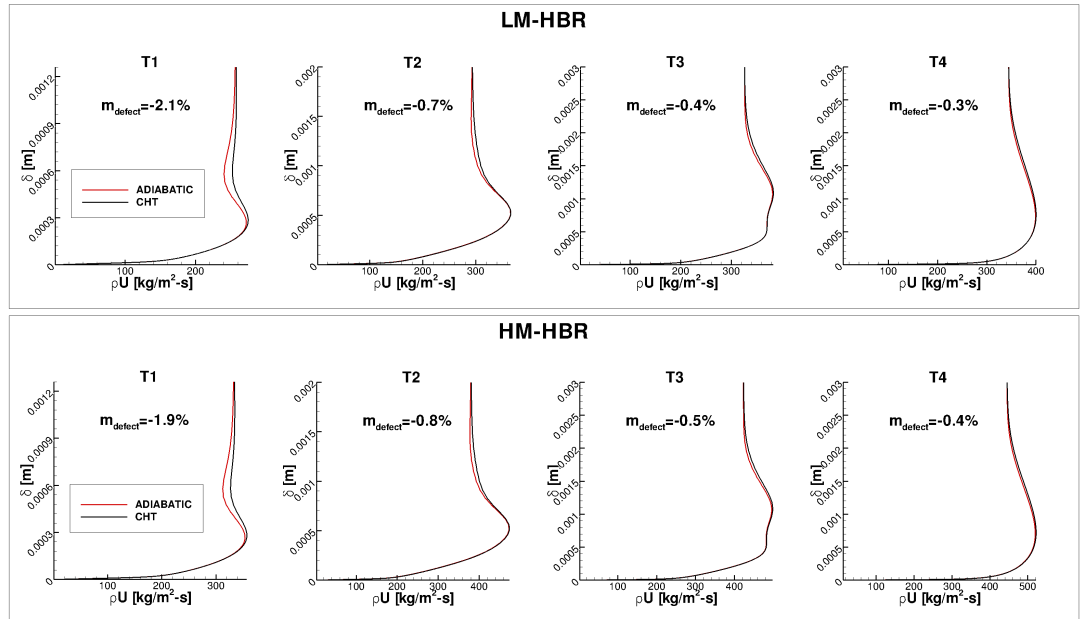


Figure 9.17: Distributions of ρU through the four traversing cuts $T1$, $T2$, $T3$, $T4$ for the $LM - HBR$ and $HM - HBR$ cases.

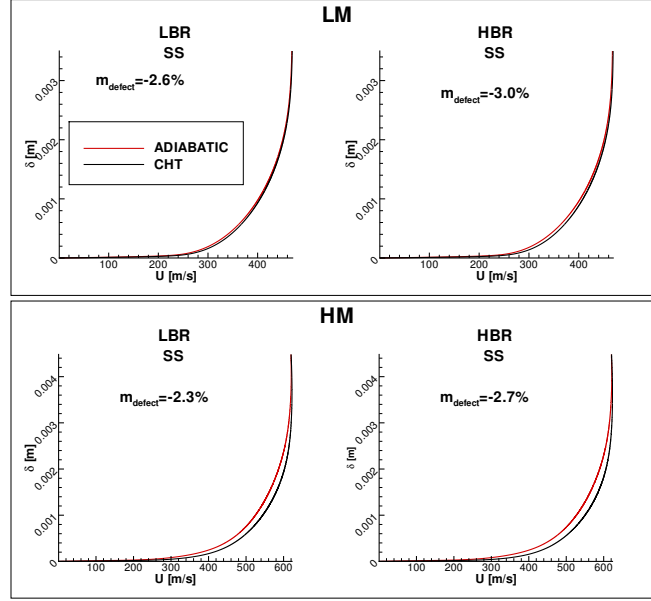


Figure 9.18: Distributions of U through the suction side traversing cut SS for the $LM - LBR$, $LM - HBR$, $HM - LBR$ and $HM - HBR$ cases.

shown.

From the results of Fig. 9.19(b) it is possible to notice that the influence of CHT on the TE shock seems to have a strong impact on the velocity distribution in the plane of the suction side traversing. The reduced static pressure of the ejected coolant at the trailing edge in the CHT case can give a first explanation of this effect. Despite the higher total pressure due to heat addition and the corresponding lower density, the increased U in the SS cut plane seems to overcompensate these effects giving a 1% reduction in static pressure of the ejected coolant in the CHT case compared to the adiabatic. This has been found in both LBR and HBR conditions.

The author believes, however, that a more detailed analysis of the effects of CHT on the trailing edge shocks for the present case cannot be carried out in the context of steady RANS. If it is true on one side that the trailing edge shock system is heavily influenced by the base pressure and coolant discharge (Sieverding (1982), Saracoglu et al. (2013)), so that a CHT effect may be effectively observed, it is also true that flow physics at the trailing edge is dominated by unsteady phenomena that steady

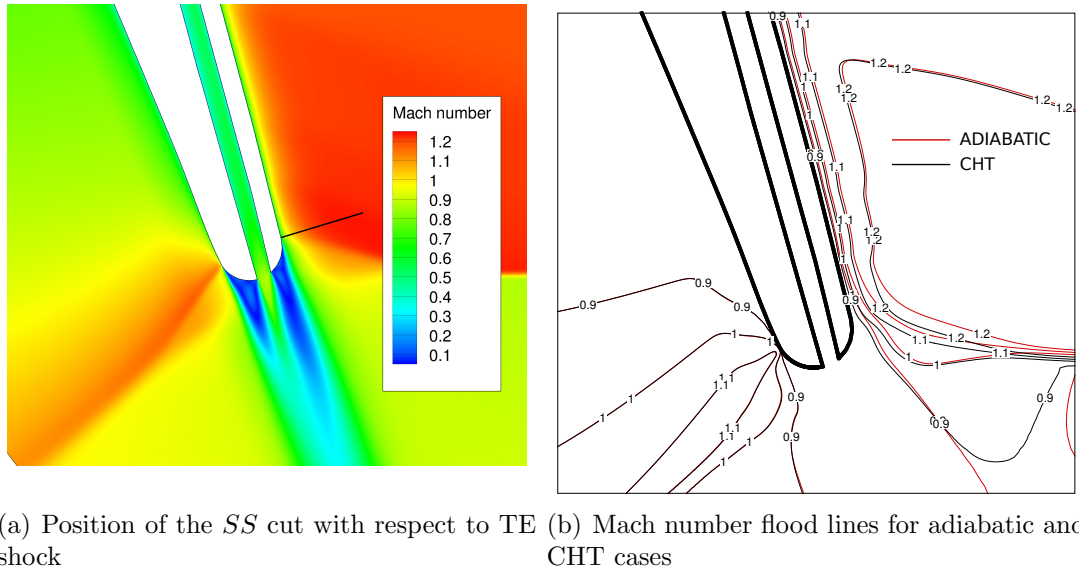


Figure 9.19: Details of the position of SS cut with respect to the TE shock and the differences between adiabatic and CHT trailing edge Mach number contours. The results shown refer to $HM-HBR$ case.

calculations cannot account for (Vagnoli et al. (2015)). On this issue, Ning & He (2001) show that the base pressure as predicted from steady RANS can be markedly different from the time-averaged base pressure obtained from unsteady calculations.

Figure 9.20 shows the ρU distribution in the SS cut plane. Differently than for what seen on the pressure side cuts, the high blowing ratio case shows the highest differences, with a mass flow defect up to 3% for the LM case, in the high blowing ratio conditions. This is obvious as to a higher coolant mass flow in the trailing edge passage corresponds an increase of wall heat flux and a higher departure from the adiabatic case. This is opposite to what happens on the film side where to higher BR correspond a bigger thermal shielding of the wall. A more heavily film cooled external surface is closer to the adiabatic behavior, while the opposite holds for internally cooled walls, where the impact of CHT is potentially higher.

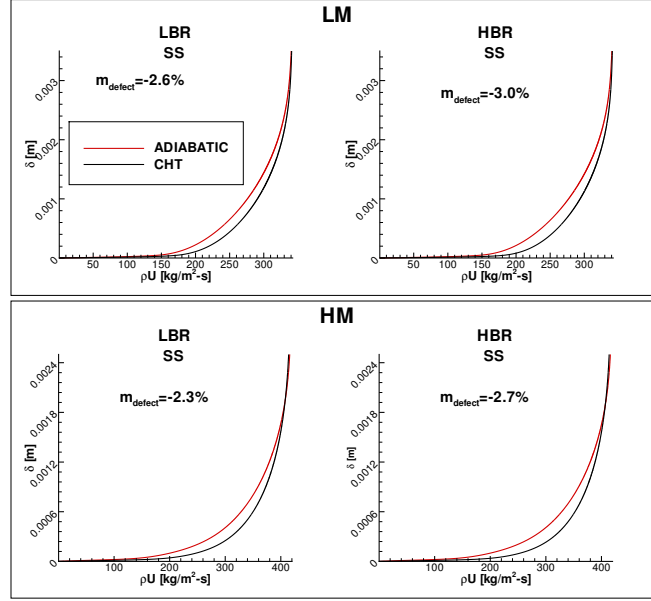


Figure 9.20: Distributions of ρU through the suction side traversing cut SS for the $LM - LBR$, $LM - HBR$, $HM - LBR$ and $HM - HBR$ cases.

9.5.3 Wall shear stress

Calculation of the wall shear stress distribution on the entire wall surface has not been possible due to post-processing limitations. A discrete data sampling in the boundary layer traversing points has been performed. The results of the analysis are shown in Fig. 9.21.

Differences in wall shear stress are negligible in all the film cooling traversing. With the higher differences at $T1$, as expected from the previous discussion on the effect of film coverage.

The late suction side point shows the highest differences, with the CHT cases for both the LM conditions showing an increase of 20% in wall shear stress with respect to the adiabatic simulations. These results suggest that effect of wall temperature on the near wall boundary layer, showing a higher near wall gradient is greater than the corresponding decrease in viscosity with the lower wall temperature, bringing to a considerably higher wall shear stress in the CHT case. This trend of increasing wall

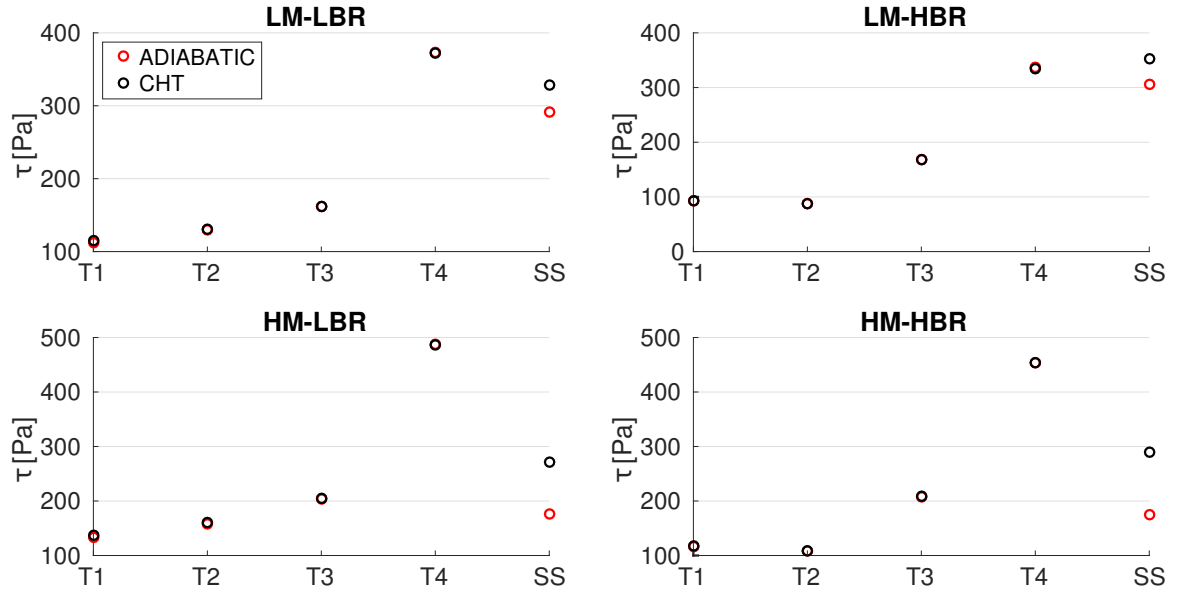


Figure 9.21: Local values of wall shear stress at the positions of boundary layer traversing $T1$, $T2$, $T3$, $T4$ and SS (see Fig. 9.13) for the $LM - LBR$, $LM - HBR$, $HM - LBR$ and $HM - HBR$ cases.

shear stress with the reduction of wall temperature is confirmed also by the results of Fig. 4.18.

The shear results at HM conditions show much higher differences in the SS shear, up to 50%. These differences are, as it is possible to infer from Fig. 9.19(b), localized to the shock region and should not be considered representative of the late suction side boundary layer.

9.6 Discussion and final remarks

The chapter focused on the evaluation of the effects of conjugate heat transfer on a double skin cooling design under matching external Biot conditions.

The effect of CHT on the film cooling discharge conditions has been examined together with external flow development through traversing in four sections on both pressure and suction surfaces.

The results show that the effects of CHT on the analyzed geometry are limited to

the first row of film cooling and to the trailing edge ejection slot: the regions more exposed to the thermal load of external gas.

The film cooled pressure side and the internally cooled suction side show opposite behavior in relation to the CHT effect on the coolant and external flow. Increased film cooling reduces the CHT effect as it increases the thermal shielding of the wall, while increased internal cooling leads the wall condition to depart more from the adiabatic state.

The boundary layer traversing show that, where the effect of CHT is more notable, differences in local flow density can lead to a reduction of local boundary layer displacement thickness. On the contrary the increase of near wall velocity gradient results in higher wall shear stresses. The two things are expected to have opposite effects on the blade profile losses.

Analysis of the boundary layer flow show that the ρU profiles are more sensitive to wall temperature variation than velocity profiles alone. Though small, the effect of wall temperature on U is such that the near wall velocity gradient is altered, resulting in non-negligible differences in wall shear in regions where the CHT effect is more significant.

From the results it seems clear, however, that the analyzed cooling passages are markedly over-sized with respect to what a micro-cooling system should provide in terms of the cooling mass flow ratio. In the interest of coolant flow economy, an ideal micro-cooling system with "continuous" film cooling ejection would be designed to limit the interactions between cooling holes and this was not the case for the chosen design. Due to the cooling layout chosen for the pressure side cooling it has been impossible to analyze a configuration with lower blowing ratios, resulting in less interaction between the coolant jets, without resulting in hot gas ingestion.

Despite the results shown suggested a weak influence of CHT on film cooling performance parameters, the author believes that potentially higher differences may

be expected in a more realistic configuration with stronger flow thermal interaction between successive film cooling holes. Also, in a configuration with a "full-coverage" film ejection, the eventual added effects of the single, even small, differences in film blowing ratios may sum up to change the total cooling mass flow ratio by a non-negligible amount.

Chapter 10

Conclusions

10.1 Effect of T_w on HTC and aerodynamics

In the first part of this thesis the effect of wall temperature on the convective heat transfer coefficient and the link between aerodynamics and heat transfer in high pressure turbines has been investigated through computational analyses using steady RANS both on 2D and 3D NGV configurations.

The results highlight the coupling existing between the aerodynamic and thermal fields. Effects of the wall temperature on shock displacement, secondary flows, wall shear, transition point and passage mass flow rate have been observed.

Variations of HTC with wall temperature have been studied using a conventional two-point method as a finite-difference operator to obtain local values of HTC at a defined temperature. The external heat transfer coefficient showed non-uniform spatial dependence, highlighting the connection between local flow history and the heat transfer coefficient. The effect of T_w has been shown to increase in prominence as the boundary layer develops on the blade surface. In a 3D flow the regions dominated by secondary flows showed to have a higher dependence of HTC on wall temperature. The introduction of non-fixed transition and its sensitivity on wall temperature for

state of the art transition models, have been shown to exacerbate the limitations of a linear scaling for heat transfer.

10.2 Non-linear HTC scaling method

The shown connection between aerodynamics and heat transfer and the fact that HTC does depend on the thermal boundary condition through the local flow history, does question the conventional approach that the Nusselt number can be scaled globally with respect to the wall to freestream temperature ratio (TR), accounting only for the variations of air properties at different temperatures.

The conventional approach of uniform scaling of Nu with TR has been shown to be inadequate in accounting for the local dependence of HTC on T_w . As a result the author proposed a new method of scaling the wall heat flux accounting for a non-linear variation of wall heat flux with wall temperature. The method can be seen as a three-point curve fitting of heat flux curve and represents a fundamental departure from the commonly used two-point method. If the wall temperature has an effect on HTC, the two-point method can be used only to obtain values of h at a specific temperature: the three-point method allows instead to scale the heat flux over a range of wall temperatures.

The proposed three-point method proved to be more accurate in scaling the heat flux at different wall temperatures with respect to the usual assumption of a linear behavior of Newton's law of cooling. It has also been shown that the adoption of such method could be beneficial for the stability of loosely coupled iterative methods where transition is present and a traditional two-point method scaling would result in non-physical values for HTC. The method has been recently adopted by other authors to account for HTC dependence with T_w (see Zhang & He (2014), Liu et al. (2015), Liu et al. (2014), Lavagnoli et al. (2014), Caloni & Shahpar (2015)) and

has been integrated in Rolls-Royce thermal analysis methods, proving to be more effective than a traditional linear scaling of the heat flux in loosely coupled conjugate calculations (Caloni & Shahpar (2015)).

Despite the improvement given by a non-linear scaling tool for the wall heat flux, the acknowledgement that the local flow history does impact the HTC calls directly for the need of Conjugate Heat Transfer (CHT) if accurate temperature predictions are sought.

10.3 Implementation and development of a CHT simulation environment

For this reason fully-coupled steady CHT capabilities have been developed and implemented in Rolls-Royce Hydra CFD solver. Since the CHT implementation is fully-coupled, the solid and the fluid domains are simultaneously marched in time in the same solver, handling both domains at the same time. This requires no iterations between numerical solution of the solid and fluid domains alone with continuous exchange of boundary conditions between the two. The fully-coupled implementation ensures also the unconditional stability of the solver. The interface between the solid and the fluid is handled internally by the solution process ensuring conservation of energy and continuity of temperature.

Implementation of the CHT capabilities required the development of a pre-processor to manipulate the solid and the fluid grids and generate the relevant mesh connectivity, allowing for a coupled solution. The implemented pre-processing tool merges the solid and the fluid grids, connects the interface points and retrieves for each node at the fluid-solid interface its connected near-wall node, needed for the CHT coupling. Due to the nodal based data structure of Hydra finite volume method, the coupling strategy and the pre-processor have been designed to handle also wall nodes that

do not have a direct connection through an edge with near-wall nodes. Such nodes, referred to as corner nodes, are treated separately from the other fluid-solid interface nodes in the CHT update process. Such differentiation between corner nodes and other CHT wall nodes has been showed to have no detrimental effect on the unconditional stability of the solver.

Constant heat flux and convective boundary conditions have been added to the solver, including a fixed point transition model with ramping of the turbulent viscosity in the transition region. Such implementations were needed to have the possibility of running simulations under more realistic boundary conditions than originally available, and to correctly model the wall heat flux in the transition region typified in the case of the NASA C3X NGV used for validation.

10.4 CHT solver validation

The implemented boundary conditions have been validated against flat plate solutions and CFD solutions obtained with a well established commercial software. The predictions of wall heat flux using the developed fixed-transition model have been validated against experimental data for transitional boundary layer on flat plate.

The modifications made to Hydra to handle the thermal solution of the solid domain have been validated against analytical solution of a thermal field in a hollow cylinder and against FEM solution of a L-shaped beam under a temperature gradient. The validation results showed a perfect match between the obtained results and the reference cases.

Validation of the CHT solver has been performed against Fluent results. Two cases have been taken as a reference for validation: flat plate flow and a typical NGV flow. In the flat plate cases both laminar and turbulent flows were examined. The bottom wall of the flat plate has been subject to: constant temperature, constant heat

flux and convective boundary conditions. For all the analyzed cases the predicted wall temperatures were in agreement with Fluent predictions by less than 1% error.

The validation test on a NGV flow has been performed against conjugate experimental data on the NASA C3X NGV. Considering the inherent experimental and numerical uncertainties a very good agreement has been achieved, comparable with other validation results available in the open literature on the same case.

10.5 CHT evaluation of a double-wall trailing edge cooling system

The implemented solver has been used for a CHT evaluation of a double-skin trailing edge cooling design under matching external Biot number conditions. This was done to analyze the possible effect of CHT on the coolant and external flow field with respect to adiabatic conditions. The design has been analyzed under two blowing ratio conditions and considering high subsonic and transonic external flow cases.

The proposed cooling layout for the trailing edge has 3 film cooling ejection holes on the pressure side and an internal cooling channel with trailing edge ejection for the suction side. The results showed that the effects of CHT are prominent where the shielding of the film is lower. The regions that are more exposed to the thermal loading of external flow (the first row of cooling holes and the trailing edge slot) showed to have higher differences in coolant discharge characteristics as a consequence of coolant heat pick-up.

As a consequence of this, the external flows on the pressure surface and suction surface sides showed to have opposite behaviors concerning the effect of an increased blowing ratio in the respective cooling channels. An increase of the film blowing ratio caused the external flow conditions to approximate more closely the adiabatic state. The opposite has been observed for the internal suction side cooling, where increased

blowing ratio increased the CHT effect on external flow.

Traversing of the external flow at different positions has been performed, showing that where the effect of CHT is prominent, differences in local fluid density can lead to a reduction of local boundary layer displacement thickness. This is expected to have a beneficial effect on profile losses. On the other hand the different flow density is shown to have an impact on external aerodynamics, increasing the near wall velocity gradient and resulting in higher wall shear stresses, with a negative impact on losses.

It is interesting to note that to a cooler wall corresponds a higher shear stress despite the reduction of viscosity with temperature. This supports one of the key points of this thesis: that the wall temperature impacts the aerodynamics and the effects of the wall thermal boundary condition should not be accounted for by just considering the variation of fluid properties with temperature.

10.6 Future works

The computations carried out for the first part of this thesis have been performed using state of the art, widely validated, commercial software. All the relevant numerical sensitivity studies have been performed, aiming to assess the reliability of the calculations in the context of a typical RANS environment. This said, as pointed out in Sec. 2.3, the accuracy of numerical simulations of turbomachinery flows, specially when heat transfer is of interest, is strongly dependent on the reliability of the considered turbulence/transition model and the hypotheses they are based on.

One of the key issues with traditional RANS modeling (common to the studies performed in this work) is the assumption of a constant turbulent Prandtl number. This assumption has been shown to be not always valid (see Sec.2.3) and certainly contributes to the general uncertainty of heat flux modeling in RANS.

Transition modeling, despite the improvements of latest developed turbulence

models, is far from being a tool of general validity. This is particularly true if the effect of wall temperature on bypass transition is investigated: an issue on which very scarce, if not any, knowledge exists in the first place and for which currently available transition models have not been validated against.

The author believes the issue of HTC dependency with wall temperature in high pressure turbines is of great engineering relevance and strongly hopes the present contribution will contribute to the general awareness of the scientific community on the issue. Particularly, considering the shortcomings inherent to RANS turbulence/transition modeling relevant to the issue, the author would like to emphasize the importance of well-conditioned experimental data to back up and validate the findings of this work. These works should go both in the direction of confirming the observed non-linearities of wall heat flux with wall temperature and deepening the understanding of the aero-thermal interaction in the boundary layer due to wall thermal boundary condition, including the effects of T_w on bypass transition.

Appendix A

The CHT pre-processor

A.1 The CHT preprocessor

The CHT coupling strategy and the treatment of transition zones requires to extract from the mesh additional connectivity information not available after execution of the standard Hydra mesh preprocessing tools. The following sections describe the way the implemented preprocessor manipulates the grid to map the CHT interface and the transition region.

The pre-processor is designed to work only with 3D grids with hexahedral elements.

A.1.1 Generation of the CHT coupling connectivity

The workflow of the preprocessor is illustrated in Fig. A.1. Generation of the CHT connectivity is achieved through the following steps:

1. Merging of solid and fluid grids in a single file
2. Identifying coincident nodes on the fluid and solid sides of the CHT boundaries and linking them together
3. Marking CHT nodes as *planar* or *corner*

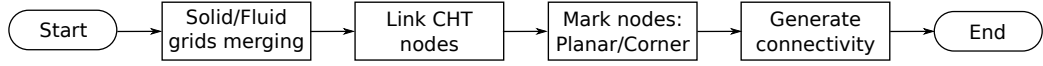


Figure A.1: Workflow of the preprocessor for the generation of CHT coupling connectivity.

4. Generating connectivity information for temperature update in planar nodes
5. Generating connectivity information for temperature update in corner nodes
6. Generating connectivity information for heat flux/convective walls

Going through the details of each of the previously mentioned steps is superfluous, the only step that will be described is the one relative to the mapping of corner and planar nodes.

A.1.1.1 Identification of planar and corner nodes

Once all the nodes belonging to the CHT interface are marked the pre-processor identifies, for each of them, whether it is a planar node or a corner node.

The process is completely automated and is based on both grid topology and the type of boundary conditions applied. The situation is better explained with the help of Fig. A.2. The figure represents boundary hexas belonging to the solid grid at the CHT interface. The matching hexas on the fluid side of the CHT wall are not shown for the sake of clarity. Nodes are tagged based on the number N of hexas that are sharing the same node. Figure A.2 shows all the possible conditions with their relative tag as corner or planar nodes.

The number of hexas adjacent to the node is, however, not sufficient to determine whether a node should be tagged as a corner node or a planar node. Figure A.2 shows that, in the case of $N = 4$ the node can belong to a corner or can be internal to a planar surface.

Nodes with $N = 1$ and $N = 2$ should normally be tagged as corner nodes, but must be considered as planar nodes in case local 1D conduction is of interest (e.g. edges and corners of the wall in the flat plate validation cases shown in Ch. 8).

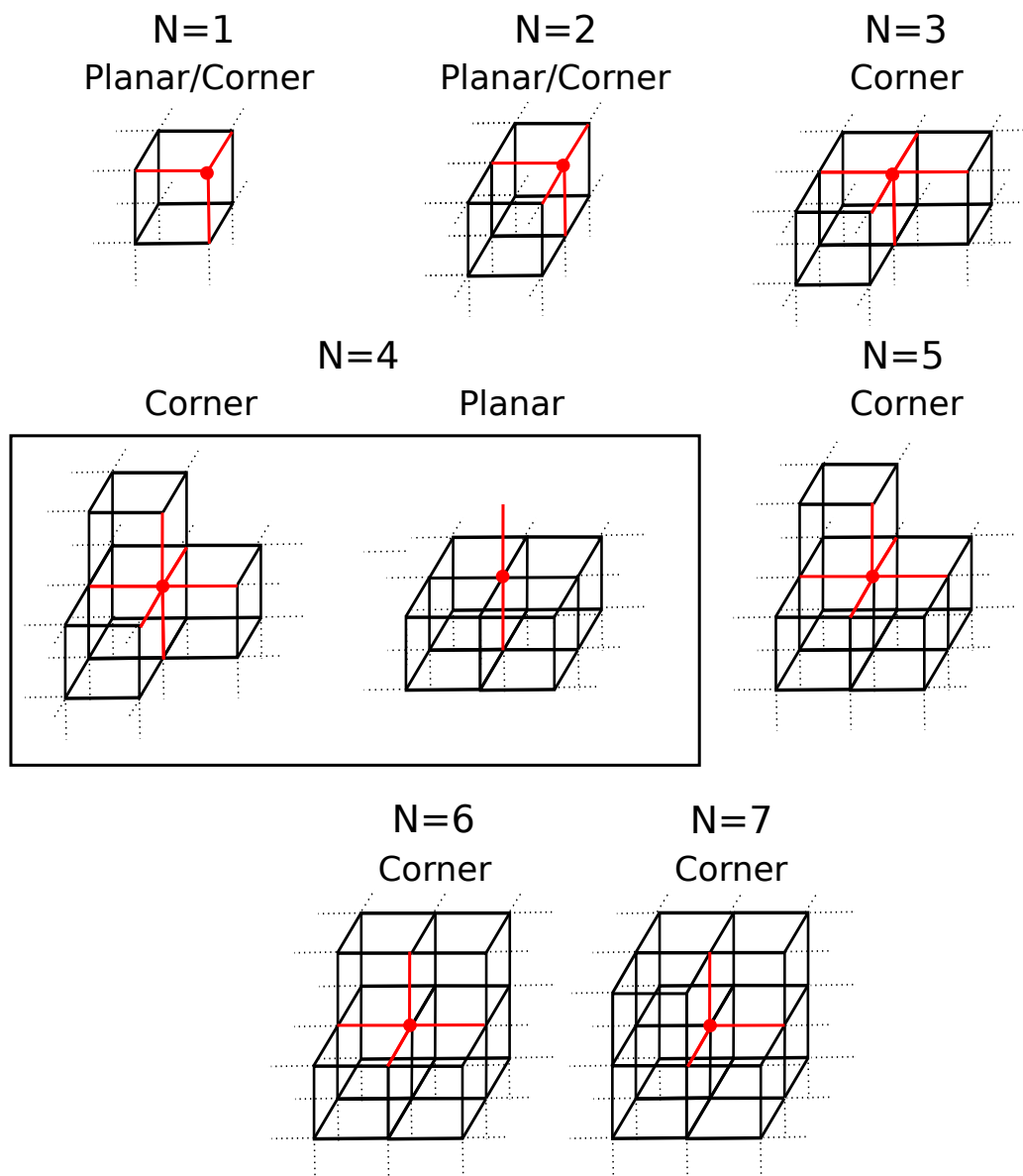


Figure A.2: Wall planar and corner nodes mapping based on number of hexas sharing the same node N . Red edges connect the relevant nodes used for wall temperature update in the CHT solution.

The pre-processor is designed to use information on both grid topology and the boundaries types to distinguish between the various cases and categorize the nodes accordingly. For each node then the relevant near-by nodes are identified (connected by red edges in Fig. A.2) to update the wall temperature during CHT calculations according to the update strategy described in Sec. 7.3.

For each planar node the pre-processor marks the near-wall nodes on both fluid and solid side. For corner nodes all the nearby nodes lying on the wall are extracted. Such nodes constitute the E_j set of nodes in Eqn. (7.5).

A.1.2 Mapping of the transition region

The mapping of the transition region allows to tag the nodes belonging to the laminar and transitional sub-domains as described in Sec. 7.5.1. The approach used consists on using edge connectivity data to perform the mapping along grid lines, as one would normally proceed with a structured mesh.

Particularly in each node a local frame of reference is built, allowing to identify a local *spanwise* $\vec{s}$ and *normal* $\vec{n}$ direction.

The mapping algorithm used has been chosen as it is insensitive to mesh skewness and curvature and requires minimum input from the user: a starting point P , the wall interested by the boundary layer transition (*transition wall* in the following) and the length and normal extension of laminar and transitional zone. The mapping process will be described using Fig. A.3.

The process starts by looking at the closest node to point P situated on a corner between the transition wall and another boundary (node 1 in Fig.A.3(a)).

Before moving to the next node in the spanwise direction (node 2), information on local wall normal is used to tag all nodes along the $\vec{n}$ direction. The mapping proceeds in the wall-normal direction by selecting, for each node, the near-by node that maximizes the dot product with the $\vec{n}$ unitary vector. The normal direction is updated in

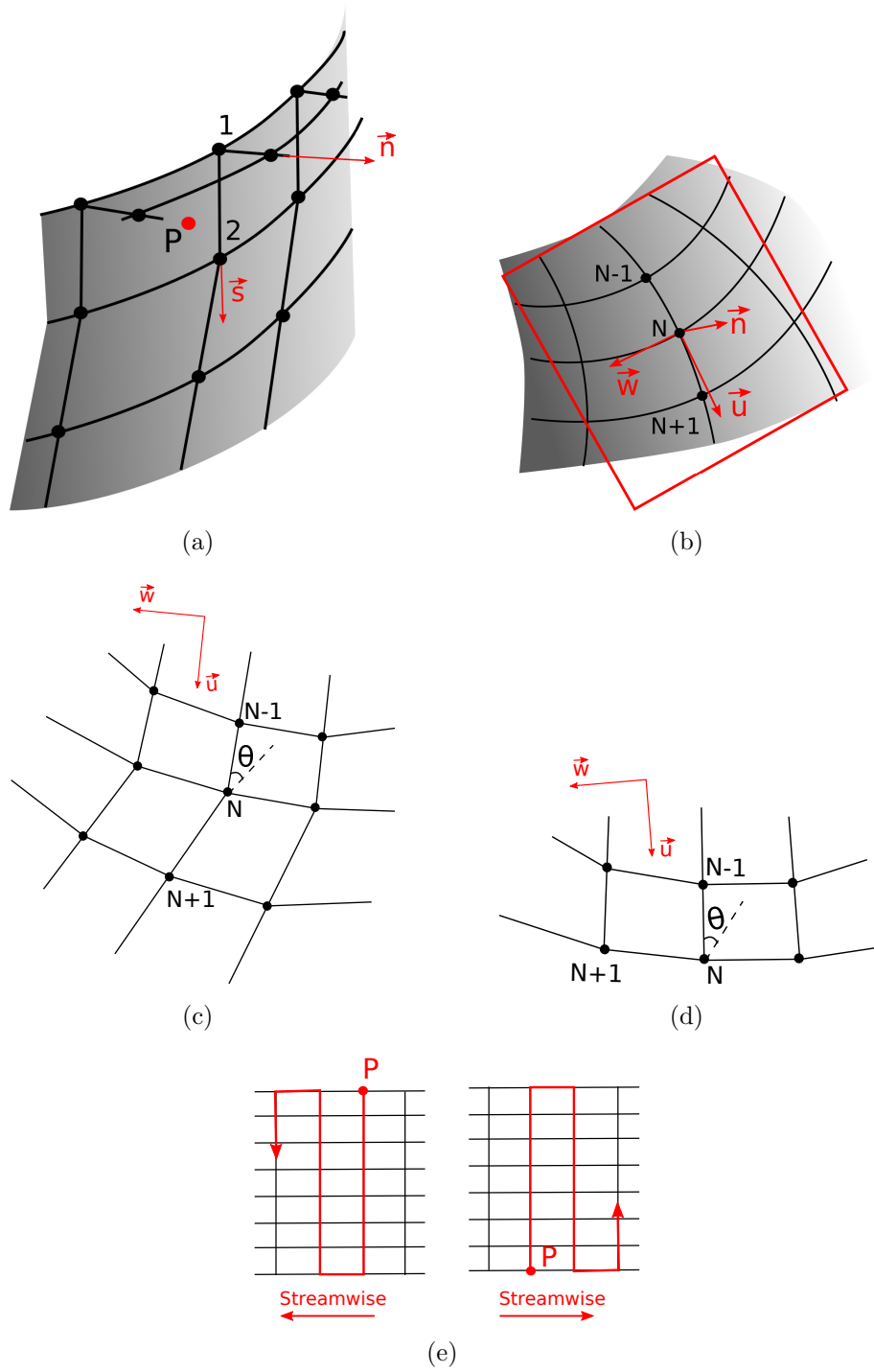


Figure A.3: Mapping of the transition region.

each node based on the previous normal node mapped.

The approach used for mapping nodes in the normal direction is, however, quite sensitive to the skewness of the grid. In skewed grids the use of the maximum dot product as a mapping criterium can result in the code follow the wrong mapping direction. Due to his skewness sensitivity this method has only been used to map nodes along $\vec{n}$ as they are all confined in the boundary layer grid, where skewness is normally controlled.

After mapping all the nodes along $\vec{n}$ the program moves to the next spanwise node. The identification of the spanwise direction makes use of a different strategy than the maximization of dot product.

For the first node the spanwise direction $\vec{s}$ is determined easily as only node 2 is not on the edge of the transition wall. In a generic internal node N on the transition wall (A.3(b)), the streamwise direction is determined as follows:

1. A plane passing by node N and with normal $\vec{n}$ is created. Streamwise vector $\vec{s} = \frac{\vec{x}(N) - \vec{x}(N-1)}{|\vec{x}(N) - \vec{x}(N-1)|}$ is projected on the plane such that:

$$\vec{u} = \vec{s} - (\vec{s} \cdot \vec{n})\vec{n}$$

is the projection of $\vec{s}$ on the plane. A third vector $\vec{w} = \vec{u} \times \vec{n}$ allows to define a local frame of reference for node N (Fig. A.3(b)).

2. The coordinates of all nodes connected to node N are projected on the $\vec{n}$ plane. The 2D (u, w) space is now considered.
3. A further coordinate transformation from cartesian (u, w) , to cylindrical (r, θ) is performed. The angle θ is measured from the direction $\vec{x}(N) - \vec{x}(N-1)$.
4. Near-by nodes are ranked by growing values of theta. The third node (the first being always, by definition of θ , node $N-1$) is selected as the next spanwise

node $N + 1$ (Fig. A.3(c)).

Using a local cylindrical coordinate system allows to avoid mapping problems due to skewness. Once the mapping in the spanwise direction reaches the wall on the other side from the starting point, the mapping advances in the streamwise direction. The successive wall point $N + 1$ is still taken as the third, in terms of growing values of θ (Fig. A.3(d)). The used algorithm does not need a special treatment for advancing in the streamwise direction or in the spanwise. Also the fact that the third node is always selected for advancing in the streamwise direction has, as a consequence, the fact that the position of the starting point P fully determines the streamwise direction of the transition zone mapping (Fig. A.3(e)). This simplifies the inputs needed and makes the mapping process much robust.

Bibliography

- Agostini, F. & Arts, T. (2005), Conjugate heat transfer investigation of rib-roughened cooling channels, *in* ‘ASME Turbo Expo 2005: Power for Land, Sea, and Air’, American Society of Mechanical Engineers, pp. 239–247.
- Albert, J. E. & Bogard, D. G. (2013), ‘Measurements of adiabatic film and overall cooling effectiveness on a turbine vane pressure side with a trench’, *Journal of Turbomachinery* **135**(5), 051007.
- Albert, J. E., Bogard, D. G. & Cunha, F. (2004), Adiabatic and overall effectiveness for a film cooled blade, *in* ‘ASME Turbo Expo 2004: Power for Land, Sea, and Air’, American Society of Mechanical Engineers, pp. 251–259.
- Amaral, S., Verstraete, T., Van den Braembussche, R. & Arts, T. (2010), ‘Design and optimization of the internal cooling channels of a high pressure turbine blade part i: Methodology’, *Journal of Turbomachinery* **132**(2), 021013.
- Anderson, A. M. & Moffat, R. J. (1992), ‘The adiabatic heat transfer coefficient and the superposition kernel function: Part 1 data for arrays of flatpacks for different flow conditions’, *Journal of Electronic Packaging* **114**(1), 14–21.
- Andrews, S. & Bradley, P. (1957), Heat transfer to turbine blades, Technical Report CP No. 294.
- Ansys, A. F. (2011), ‘14.0 theory guide’, *ANSYS inc* pp. 218–221.

- Back, L., Cuffel, R. & Massier, P. (1969), ‘Laminar, transition, and turbulent boundary-layer heat-transfer measurements with wall cooling in turbulent airflow through a tube’, *Journal of Heat Transfer* **91**(4), 477–487.
- Barnes, J. (1960), An experimental investigation of heat transfer from the inside surface of a hot smooth tube to air, helium and carbon dioxide, Reports and Memoranda 3246, Aeronautical Research Council.
- Barth, T. (1994), Aspects of unstructured grids and finite volume solvers for the Euler and Navier-Stokes equations, in ‘VKI Lecture Series’, von Karman Institute for Fluid Dynamics.
- Bergeles, G., Gosman, A. & Launder, B. (1978), ‘The turbulent jet in a cross stream at low injection rates: a three-dimensional numerical treatment’, *Numerical Heat Transfer, Part A: Applications* **1**(2), 217–242.
- Blazek, J. (2005), *Computational Fluid Dynamics: Principles and Applications: (Book with accompanying CD)*, Elsevier.
- Bohn, D., Ren, J. & Kusterer, K. (2003), Conjugate heat transfer analysis for film cooling configurations with different hole geometries, number ASME paper GT2003-38369.
- Boyce, M. P. (2012), *Gas Turbine Engineering Handbook (Fourth Edition)*, fourth edition edn, Butterworth-Heinemann.
- URL:** <http://www.sciencedirect.com/science/article/pii/B9780123838421000111>
- Brandt, A. (1980), ‘Multilevel adaptive computations in fluid dynamics’, *AIAA Journal* **18**(10), 1165–1172.
- Brown, W. B. & Donoughe, P. L. (1950), Extension of boundary-layer heat-transfer theory to cooled turbine blades, Technical Report NACA-RM-E50F02.

- Bunker, R. S. (2013), Gas turbine cooling: Moving from macro to micro cooling, *in* ‘ASME Turbo Expo 2013: Turbine Technical Conference and Exposition’, American Society of Mechanical Engineers, pp. V03CT14A002–V03CT14A002.
- Bunker, R. S., Bailey, J. C., Lee, C.-P. & Stevens, C. W. (2004), In-wall network (mesh) cooling augmentation of gas turbine airfoils, *in* ‘ASME Turbo Expo 2004: Power for Land, Sea, and Air’, American Society of Mechanical Engineers, pp. 1007–1018.
- Caloni, S. & Shahpar, S. (2015), Investigation into coupling techniques for a high pressure turbine blade tip, *in* ‘ASME Turbo Expo 2015: Turbine Technical Conference and Exposition’, American Society of Mechanical Engineers, pp. V05AT10A017–V05AT10A017.
- Chana, K. S., Patel, T. & Mole, A. H. (2001), A summary of measurements with a non uniform inlet temperature profile from the MT1 single stage HP turbine, Technical Report DERA/AS/PPD/CR010116, QinetiQ.
- Chi, X. & Shih, T. I. (2012), ‘Bulk temperature, heat-transfer coefficient, and nusselt number–revisited’, *AIAA Paper* **807**.
- Chorin, A. (1968), ‘Numerical Solution of the Navier-Stokes Equations’, *Mathematics of Computation* **22**, 745–762.
- Coletti, F., Scialanga, M. & Arts, T. (2010), Experimental investigation of conjugate heat transfer in a rib-roughened trailing edge channel with crossing-jets, number ASME paper GT2010-22432.
- Cukurel, B. & Arts, T. (2013), ‘Local heat transfer dependency on thermal boundary condition in ribbed cooling channel geometries’, *Journal of Heat Transfer* **135**(10), 101001.

- Cukurel, B., Arts, T. & Selcan, C. (2012), ‘Conjugate heat transfer characterization in cooling channels’, *Journal of Thermal Science* **21**(3), 286–294.
- Dawes, W. N., Kellar, W. P. & Harvey, S. A. (2010), Towards cooled turbine preliminary life prediction via concurrent aerodynamic, thermal and material stress simulations on conjugate meshes, number ASME paper GT2010-22482.
- Day, C., Oldfield, M. & Lock, G. (2000), ‘Aerodynamic performance of an annular cascade of film cooled nozzle guide vanes under engine representative conditions’, *Experiments in fluids* **29**(2), 117–129.
- Dees, J. E., Bogard, D. G., Ledezma, G. A. & Laskowski, G. M. (2011), The effects of conjugate heat transfer on the thermal field above a film cooled wall, number ASME paper GT2011-46617.
- Denton, J. D. (2010), Some limitations of turbomachinery cfd, *in* ‘ASME Turbo Expo 2010: Power for Land, Sea, and Air’, American Society of Mechanical Engineers, pp. 735–745.
- Denton, J. & Xu, L. (1990), ‘The trailing edge loss of transonic turbine blades’, *Journal of Turbomachinery* **112**(2), 277–285.
- Dittus, F. & Boelter, L. (1985), ‘Heat transfer in automobile radiators of the tubular type’, *International Communications in Heat and Mass Transfer* **12**(1), 3–22.
- Dobrowolski, L., Bogard, D. & S., R. (2010), ‘Accuracy of conventional adiabatic effectiveness and heat transfer augmentation factors in predicting heat flux into a turbine blade leading edge’, *ASME Conference Proceedings* **2011**(23438).
- Dorfman, A. & Renner, Z. (2009), ‘Conjugate problems in convective heat transfer: Review’, *Mathematical Problems in Engineering* **2009**.

- Dunn, M. (2001), ‘Convective heat transfer and aerodynamics in axial flow turbines’, *Journal of Turbomachinery* **123**, 637–686.
- Dunn, M., Rae, W. & Holt, J. (1984), ‘Measurement and analyses of heat flux data in a turbine stage: Part ii discussion of results and comparison with predictions’, *Journal of Engineering for Gas Turbines and Power* **106**(1), 234–240.
- Dyson, T. E., Bogard, D. G. & Bradshaw, S. D. (2014), ‘Evaluation of cfd simulations of film cooling performance on a turbine vane including conjugate heat transfer effects’, *International Journal of Heat and Fluid Flow* **50**, 279–286.
- Eckert, E. (1955), ‘Engineering relations for friction and heat transfer to surfaces in high velocity flow’, *Journal of the Aeronautical Sciences* **22**(8), 585–587.
- Eckert, E. & Drake, R. (1971), *Analysis of heat and mass transfer*, McGraw-Hill series in mechanical engineering, McGraw-Hill.
URL: https://books.google.be/books?id=U_tQAAAAMAAJ
- Ekkad, S. V., Huang, Y. & Han, J.-C. (1999), ‘Impingement heat transfer on a target plate with film cooling holes’, *Journal of Thermophysics and Heat transfer* **13**(4), 522–528.
- Facchini, B., Magi, A. & Greco, A. S. D. (2004), ‘Conjugate heat transfer simulation of a radially cooled gas turbine vane’, *ASME Conference Proceedings* **2004**(41685), 951–961.
URL: <http://link.aip.org/link/abstract/ASMECP/v2004/i41685/p951/s1>
- Fedrizzi, R. & Arts, T. (2004), Investigation of the conjugate convective-conductive thermal behavior of a rib-roughened internal cooling channel, in ‘ASME Turbo Expo 2004: Power for Land, Sea, and Air’, American Society of Mechanical Engineers, pp. 45–53.

- Fitt, A., Forth, C., Robertson, B. & Jones, T. (1986), ‘Temperature ratio effects in compressible turbulent boundary layers’, *International journal of heat and mass transfer* **29**(1), 159–164.
- Florschuetz, L., Truman, C. & Metzger, D. (1981), Streamwise flow and heat transfer distributions for jet array impingement with crossflow, *in* ‘ASME 1981 International Gas Turbine Conference and Products Show’, American Society of Mechanical Engineers, pp. V003T09A005–V003T09A005.
- Giles, M. (1997), ‘Stability analysis of numerical interface conditions in fluid-structure thermal analysis’, *International Journal for Numerical Methods in Fluids* **25**, 421–436.
- Gomes, R. A. & Niehuis, R. (2013), The concept of adiabatic heat transfer coefficient and its application to turbomachinery, number ASME paper GT2013-94715, ASME.
- Greiner, N. J., Polanka, M. D., Rutledge, J. L. & Robertson, J. R. (2014), ‘Effect of variable properties within a boundary layer with large freestream-to-wall temperature differences’, *Journal of Engineering for Gas Turbines and Power* **136**(5), 052604.
- Han, J.-C., Dutta, S. & Ekkad, S. (2012), *Gas turbine heat transfer and cooling technology*, CRC Press.
- Harrison, K. L. & Bogard, D. G. (2008*a*), Comparison of rans turbulence models for prediction of film cooling performance, *in* ‘ASME Turbo Expo 2008: Power for Land, Sea, and Air’, American Society of Mechanical Engineers, pp. 1187–1196.
- Harrison, K. L. & Bogard, D. G. (2008*b*), Use of the adiabatic wall temperature in film cooling to predict wall heat flux and temperature, *in* ‘ASME Turbo Expo

- 2008: Power for Land, Sea, and Air', American Society of Mechanical Engineers, pp. 1197–1207.
- Harten, A. (1983), 'High resolution schemes for hyperbolic conservation laws', *Journal of computational physics* **49**(3), 357–393.
- Haselbacher, A. & Blazek, J. (2000), 'Accurate and efficient discretization of navier-stokes equations on mixed grids', *AIAA journal* **38**(11), 2094–2102.
- He, L. & Oldfield, M. (2011), 'Unsteady conjugate heat transfer modeling', *Journal of turbomachinery* **133**(3, 2011).
- Heidmann, J. D., Kassab, A. J., Divo, E. A., Rodriguez, F. & Steinthorsson, E. (2003), Conjugate heat transfer effects on a realistic film-cooled turbine vane, number ASME paper GT2003-38553.
- Heidmann, J. D., Rigby, D. L. & Ameri, A. A. (2000), 'A three-dimensional coupled internal/external simulation of a film-cooled turbine vane', *Journal of Turbomachinery* **122**(2), 348–359.
- Heselhaus, A. (1998), A hybrid coupling scheme and stability analysis for coupled solid/fluid turbine blade temperature calculation, number 98-GT-88, ASME.
- Hirsch, C. (1988), *Numerical Computation of Internal and External Flows, Vol. 1 Fundamentals of Numerical Discretization, Vol. 2 Computational Methods for Inviscid and Viscous Flows*, Wiley, Chichester.
- Horlock, J. & Denton, J. (2005), 'A review of some early design practice using computational fluid dynamics and a current perspective', *Journal of Turbomachinery* **127**(1), 5–13.
- Huang, P., Bardina, J. & Coakley, T. (1997), 'Turbulence modeling validation, testing, and development', *NASA Technical Memorandum* **110446**.

- Hubbartt, J. E. (1950), Comparison of outside-surface heat-transfer coefficients for cascades of turbine blades, Technical Report NACA-RM-E50C28.
- Humble, L. V., Lowdermilk, W. H. & Desmon, L. G. (1951), Measurements of average heat-transfer and friction coefficients for subsonic flow of air in smooth tubes at high surface and fluid temperatures, Technical report, National advisory committee for aeronautics.
- Hylton, L., Mihelc, M., Turner, E., Nealy, D. & York, R. (1983), Analytical and experimental evaluation of the heat transfer distribution over the surfaces of turbine vanes, Contract Report NASA CR 168015, NASA.
- Ieronymidis, I., Gillespie, D. R., Ireland, P. T. & Kingston, R. (2010), ‘Detailed heat transfer measurements in a model of an integrally cast cooling passage’, *Journal of Turbomachinery* **132**(2), 021002.
- Illingworth, J. B., Hills, N. J. & Barnes, C. J. (2005), ‘3d fluid-solid heat transfer coupling of an aero engine pre-swirl system’, *ASME Conference Proceedings* **2005**(47268), 801–811.
URL: <http://link.aip.org/link/abstract/ASMECP/v2005/i47268/p801/s1>
- Jones, T. (1999), ‘Theory for the use of foreign gas in simulating film cooling’, *International Journal of Heat and Fluid Flow* **20**(3), 349–354.
- Kadoya, K., Matsunaga, N. & Nagashima, A. (1985), ‘Viscosity and thermal conductivity of dry air in the gaseous phase’, *Journal of physical and chemical reference data* **14**(4), 947–970.
- Kays, W. & Crawford, M. (1980), *Convective Heat and Mass Transfer-Second edition*, Tata Mc Graw-Hill Publishing Co. Ltd.

- Kays, W. M. (1994), ‘Turbulent prandtl number where are we?’, *Journal of Heat Transfer* **116**(2), 284–295.
- Kays, W. & Nicoll, W. (1963), ‘Laminar flow heat transfer to a gas with large temperature differences’, *Journal of Heat Transfer* **85**(4), 329–338.
- Kercher, D., Sheer, R. & So, R. (1982), Short duration heat transfer studies at high free-stream temperatures, in ‘ASME 1982 International Gas Turbine Conference and Exhibit’, American Society of Mechanical Engineers, pp. V004T09A005–V004T09A005.
- Khanal, B., He, L., Northall, J. & Adami, P. (2013), ‘Analysis of radial migration of hot-streak in swirling flow through high-pressure turbine stage’, *Journal of turbomachinery* **135**(4, 2013).
- Kingery, J., Ames, F., Downs, J., Acharya, S. & Barker, B. (2015), An analysis of a deposition tolerant cooling approach for nozzle guide vanes, in ‘ASME Turbo Expo 2015: Turbine Technical Conference and Exposition’, American Society of Mechanical Engineers, pp. V05BT13A006–V05BT13A006.
- Kohli, A. & Bogard, D. G. (2005), ‘Turbulent transport in film cooling flows’, *Journal of heat transfer* **127**(5), 513–520.
- Kvasnak, W., LaFleur, R., Soechting, F., Joe, C., Moroso, J. & Hayes, D. (2003), ‘Method and apparatus for cooling a wall within a gas turbine engine’. US Patent 6,514,042.
URL: <https://www.google.com.ar/patents/US6514042>
- Lad, B. & He, L. (2013), An immersed mesh block (imb) approach for conjugate heat transfer predictions, in ‘ASME Turbo Expo 2013: Turbine Technical Conference and Exposition’, American Society of Mechanical Engineers, pp. V03BT11A007–V03BT11A007.

- Lad, B., He, L. & Romero, E. (2012), Validation of the immersed mesh block (IMB) approach against the cooled MT1 NGV application for mesh dependency studies, number ASME paper GT2012-68779.
- Lakshminarayana, B. (1995), *Fluid Dynamics and Heat Transfer of Turbomachinery*, Wiley-Interscience publication, John Wiley & Sons.
URL: <http://books.google.co.uk/books?id=v6EvNjtOJcoC>
- Land, C. C., Joe, C. & Thole, K. A. (2010), ‘Considerations of a double-wall cooling design to reduce sand blockage’, *Journal of Turbomachinery* **132**(3), 031011.
- Lauder, B. E. & Spalding, D. (1974), ‘The numerical computation of turbulent flows’, *Computer methods in applied mechanics and engineering* **3**(2), 269–289.
- Lavagnoli, S., De Maesschalck, C. & Paniagua, G. (2014), Analysis of the heat transfer driving parameters in tight rotor blade tip clearances, *in* ‘ASME Turbo Expo 2014: Turbine Technical Conference and Exposition’, American Society of Mechanical Engineers, pp. V05BT14A002–V05BT14A002.
- Liu, C.-L., Zhu, H.-R. & Bai, J.-T. (2008), ‘Effect of turbulent prandtl number on the computation of film-cooling effectiveness’, *International Journal of Heat and Mass Transfer* **51**(25), 6208–6218.
- Liu, Z., Liu, Z. & Feng, Z. (2014), ‘Unsteady analysis on the effects of tip clearance height on hot streak migration across rotor blade tip clearance’, *Journal of Engineering for Gas Turbines and Power* **136**(8), 082605.
- Liu, Z., Wang, Z. & Feng, Z. (2015), Effects of inlet swirl on hot streak migration across tip clearance and heat transfer on rotor blade tip, *in* ‘ASME Turbo Expo 2015: Turbine Technical Conference and Exposition’, American Society of Mechanical Engineers, pp. V05BT13A011–V05BT13A011.

- Lovely, D. & Haines, R. (1999), Shock detection from computational fluid dynamics results, *in* ‘Proceedings of the 14th Computational Fluid Dynamics Conference, Norfolk, VA’, pp. 296–304.
- Luikov, A. (1974), ‘Conjugate convective heat transfer problems’, *International Journal of Heat and Mass Transfer* **17**(2), 257–265.
- Luikov, A., Aleksashenko, V. & Aleksashenko, A. (1971), ‘Analytical methods of solution of conjugated problems in convective heat transfer’, *International Journal of Heat and Mass Transfer* **14**(8), 1047–1056.
- Mangani, L. & Andreini, A. (2008), ‘Application of an object-oriented cfd code to heat transfer analysis’, *ASME Conference Proceedings* **2008**(43147), 999–1011.
URL: <http://link.aip.org/link/abstract/ASMECP/v2008/i43147/p999/s1>
- Mao, J., Guo, W., Liu, Z. & Zeng, J. (2007), Experimental studies on cooling effectiveness of the double-decker air jet impingement with film outflow, *in* ‘ASME Turbo Expo 2007: Power for Land, Sea, and Air’, American Society of Mechanical Engineers, pp. 295–304.
- Martinelli, L. (1987), Calculations of viscous flows with a multigrid method, Technical report, Princeton Univ., NJ (USA).
- Matsubara, M. & Alfredsson, P. H. (2001), ‘Disturbance growth in boundary layers subjected to free-stream turbulence’, *Journal of Fluid Mechanics* **430**, 149–168.
- Mayle, R. E. (1991), ‘The 1991 igtI scholar lecture: The role of laminar-turbulent transition in gas turbine engines’, *Journal of Turbomachinery* **113**(4), 509–536.
- Mayle, R. & Schulz, A. (1996), The path to predicting bypass transition, *in* ‘ASME 1996 International Gas Turbine and Aeroengine Congress and Exhibition’, American Society of Mechanical Engineers, pp. V001T01A065–V001T01A065.

- Mazur, Z., Hernandez-Rossette, A., Garcia-Illescas, R. & Luna-Ramirez, A. (2006), ‘Analysis of conjugate heat transfer of a gas turbine first stage nozzle’, *Applied Thermal Engineering* **26**(16), 1796–1806.
- Mazzoni, C. M., Luque, S. & Rosic, B. (2015), Capabilities of thermal wall functions to predict heat transfer on the ngvs of a gas turbine with multiple can combustors, in ‘ASME Turbo Expo 2015: Turbine Technical Conference and Exposition’, American Society of Mechanical Engineers, pp. V02AT38A035–V02AT38A035.
- Medic, G. & Durbin, P. (2002), ‘Toward improved prediction of heat transfer on turbine blades’, *Journal of Turbomachinery* **124**, 187–192.
- Mee, D., Baines, N., Oldfield, M. & Dickens, T. (1992), ‘An examination of the contributions to loss on a transonic turbine blade in cascade’, *Journal of turbomachinery* **114**(1), 155–162.
- Menter, F. (1994), ‘Two-equation eddy-viscosity turbulence models for engineering applications’, *AIAA journal* **32**(8), 1598–1605.
- Menter, F. R., Langtry, R., Likki, S., Suzen, Y., Huang, P. & Völker, S. (2006), ‘A correlation-based transition model using local variables part i: model formulation’, *Journal of turbomachinery* **128**(3), 413–422.
- Moffat, R. J. (1998), ‘What’s new in convective heat transfer?’, *International Journal of Heat and Fluid Flow* **20**(19), 90–101.
- Moinier, P. (1999), Algorithm Developments for an Unstructured Viscous Flow Solver, PhD thesis, St. Hugh’s College, Oxford University.
- Montomoli, F., Adami, P., Della Gatta, S. & Martelli, F. (2004), Conjugate heat transfer modelling in film cooled blades, in ‘ASME Turbo Expo 2004: Power for Land, Sea, and Air’, American Society of Mechanical Engineers, pp. 147–156.

- Montomoli, F., Massini, M., Yang, H. & Han, J. (2012), ‘The benefit of high-conductivity materials in film cooled turbine nozzles’, *International Journal of Heat and Fluid Flow* **34**, 107–116.
- Mosaad, M. (1999), ‘Laminar forced convection conjugate heat transfer over a flat plate’, *Heat and mass transfer* **35**(5), 371–375.
- Na, S., Williams, B., Dennis, R., Bryden, K. & Shih, T.-P. (2007), Internal and film cooling of a flat plate with conjugate heat transfer, *in* ‘ASME Turbo Expo 2007: Power for Land, Sea, and Air’, American Society of Mechanical Engineers, pp. 545–554.
- Nathan, M. L., Dyson, T. E., Bogard, D. G. & Bradshaw, S. D. (2014), ‘Adiabatic and overall effectiveness for the showerhead film cooling of a turbine vane’, *Journal of Turbomachinery* **136**(3), 031005.
- Ning, W. & He, L. (2001), ‘Some modeling issues on trailing-edge vortex shedding’, *AIAA journal* **39**(5), 787–793.
- Nirmalan, N. & Hylton, L. (1990), ‘An experimental study of turbine vane heat transfer with leading edge and downstream film cooling’, *Journal of turbomachinery* **112**(3), 477–487.
- Perelman, T. (1961), ‘On conjugated problems of heat transfer’, *International Journal of Heat and Mass Transfer* **3**(4), 293–303.
- Petukhov, B. (1970), Heat transfer and friction in turbulent pipe flow with variable physical properties, Vol. 6 of *Advances in Heat Transfer*, Elsevier, pp. 503 – 564.
- Pierce, N. A. & Giles, M. B. (1997), ‘Preconditioned multigrid methods for compressible flow calculations on stretched meshes’, *Journal of Computational Physics* **136**(2), 425–445.

- Rahim, A., Khanal, B., He, L. & Romero, E. (2013), Effect of ngv lean under influence of inlet temperature traverse, *in* ‘ASME Turbo Expo 2013: Turbine Technical Conference and Exposition’, American Society of Mechanical Engineers, pp. V06AT36A017–V06AT36A017.
- Ramachandran, S. G. & Shih, T. I. (2015), ‘Biot number analogy for design of experiments in turbine cooling’, *Journal of Turbomachinery* **137**(6), 061002.
- Reshotko, E. & Tumin, A. (2004), ‘Role of transient growth in roughness-induced transition’, *AIAA journal* **42**(4), 766–770.
- Reynolds, W., Kays, W. & Kline, S. (1958), Heat transfer in the turbulent incompressible boundary layer - i constant wall temperature, NASA Memorandum 12-1-58W, NASA.
- Rhee, J. & Moffat, R. J. (2006), ‘Experimental estimate of the continuous one-dimensional kernel function in a rectangular duct with forced convection’, *Journal of Heat Transfer* **128**, 811–818.
- Rigby, D. L. & Lepicovsky, J. (2001), Conjugate heat transfer analysis of internally cooled configurations, *in* ‘ASME Turbo Expo 2001: Power for Land, Sea, and Air’, American Society of Mechanical Engineers, pp. V003T01A084–V003T01A084.
- Roe, P. L. (1981), ‘Approximate riemann solvers, parameter vectors, and difference schemes’, *Journal of computational physics* **43**(2), 357–372.
- Rolls-Royce (2015), *The Jet Engine*, John Wiley & Sons.
- Rued, K. & Wittig, S. (1986), ‘Laminar and transitional boundary layer structures in accelerating flow with heat transfer’, *Journal of turbomachinery* **108**(1), 116–123.
- Saracoglu, B., Paniagua, G., Sanchez, J. & Rambaud, P. (2013), ‘Effects of blunt

- trailing edge flow discharge in supersonic regime’, *Computers & Fluids* **88**, 200–209.
- Savill, A. (1993), ‘Some recent progress in the turbulence modelling of by-pass transition’, *Near-wall turbulent flows* pp. 829–848.
- Savill, A. (1996), One-point closures applied to transition, *in* ‘Turbulence and transition modelling’, Springer, pp. 233–268.
- Schlichting, H. (1978), *Boundary Layer Theory*, 7 edn, McGraw-Hill.
- Shih, T.-H., Liou, W., Shabbir, A., Yang, Z. & Zhu, J. (1994), ‘A new k-epsilon eddy viscosity model for high reynolds number turbulent flows: Model development and validation’, *NASA STI/Recon Technical Report N* **95**, 11442.
- Sieverding, C. (1982), The influence of trailing edge ejection on the base pressure in transonic turbine cascades, *in* ‘ASME 1982 International Gas Turbine Conference and Exhibit’, American Society of Mechanical Engineers, pp. V001T01A021–V001T01A021.
- Spalart, P. & Allmaras, S. (1992), ‘A one-equation turbulence model for aerodynamic flows’, *AIAA Paper 1992-0439* **1**(2), 5–21.
- Starke, C., Janke, E., Hofer, T. & Lengani, D. (2008), ‘Comparison of a conventional thermal analysis of a turbine cascade to a full conjugate heat transfer computation’, *ASME Conference Proceedings* .
- Stewart, W. L. (1955), Analysis of two-dimensional compressible-flow loss characteristics downstream of turbomachine blade rows in terms of basic boundary-layer characteristics, Technical Report NACA-TN-3515, NACA.
- Stewart, W. R. & Bogard, D. G. (2015), ‘Experimental thermal field measurements

- of film cooling above the suction surface of a turbine vane', *Journal of Engineering for Gas Turbines and Power* **137**(10), 102604.
- Sun, Z., Chew, J. W., Hills, N. J., Volkov, K. N. & Barnes, C. J. (2008), 'Efficient fea/cfd thermal coupling for engineering applications', *ASME Conference Proceedings* **2008**(43147), 1505–1515.
URL: <http://link.aip.org/link/abstract/ASMECP/v2008/i43147/p1505/s1>
- Talib, A. R. A. (2003), Detailed Investigation Of The Low Temperature Analogy Of An Aircraft Engine Standard Fire Test, PhD thesis, University Of Oxford, Dept. Of Engineering Science.
- Tallman, J., Hadelman, C., Dunn, M., Tolpadi, A. & Bergholz, R. (2009), 'Heat transfer measurements and predictions for a modern, high-pressure, transonic turbine, including endwalls', *Journal of Turbomachinery* **131**(2).
- Tolpadi, A. K., Tallman, J. A. & El-Gabry, L. (2005), 'Turbine airfoil heat transfer predictions using cfd', *ASME Conference Proceedings* **2005**(47268), 79–86.
URL: <http://link.aip.org/link/abstract/ASMECP/v2005/i47268/p79/s1>
- Vagnoli, S., Verstraete, T., Mateos, B. & Sieverding, C. H. (2015), 'Prediction of the unsteady turbine trailing edge wake flow characteristics and comparison with experimental data', *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy* **229**(5), 487–497.
- Van Leer, B. (1979), 'Towards the ultimate conservative difference scheme. v. a second-order sequel to godunov's method', *Journal of computational Physics* **32**(1), 101–136.
- VanFossen, G. J. (1982), 'Heat transfer coefficients for staggered arrays of short pin fins', *Journal of Engineering for Gas Turbines and Power* **104**(2), 268–274.

- Verstraete, T. (2008), Multidisciplinary Turbomachinery Component Optimization Considering Performance, Stress, and Internal Heat Transfer, PhD thesis, Universiteit Gent. ISBN 978-2-930389-33-8.
- Verstraete, T., Alsalihi, Z. & den Braembussche, R. A. V. (2007), ‘Numerical study of the heat transfer in micro gas turbines’, *Journal of Turbomachinery* **129**(4), 835–841.
- Verstraete, T., Amaral, S., den Braembussche, R. V. & Arts, T. (2010), ‘Design and optimization of the internal cooling channels of a high pressure turbine blade-part II: Optimization’, *Journal of Turbomachinery* **132**(2), 021014–1–9.
URL: <http://link.aip.org/link/?JTM/132/021014/1>
- Verstraete, T. & Van den Braembussche, R. (2009), A novel method for the computation of conjugate heat transfer with coupled solvers, in ‘Int. Symp. on Heat Transfer in Gas turbine Systems, 9-14 August 2009, Antalaya, Turkey’.
- Walters, D. K. & Cokljat, D. (2008), ‘A three-equation eddy-viscosity model for reynolds-averaged navier–stokes simulations of transitional flow’, *Journal of Fluids Engineering* **130**(12), 121401.
- Walters, D. & Leylek, J. (1997), ‘A systematic computational methodology applied to a three-dimensional film-cooling flowfield’, *Journal of Turbomachinery* **119**(4), 777–785.
- Wilcox, D. C. et al. (1998), *Turbulence modeling for CFD*, Vol. 2, DCW industries La Canada, CA.
- Williams, R. P., Dyson, T. E., Bogard, D. G. & Bradshaw, S. D. (2014), ‘Sensitivity of the overall effectiveness to film cooling and internal cooling on a turbine vane suction side’, *Journal of Turbomachinery* **136**(3), 031006.

- Wlassow, F., Duchaine, F., Leroy, G. & Gourdain, N. (2010), ‘3d simulation of coupled fluid flow and solid heat conduction for the calculation of blade wall temperature in a turbine stage’, *ASME Conference Proceedings* **2010**(44021), 723–734.
URL: <http://link.aip.org/link/abstract/ASMECP/v2010/i44021/p723/s1>
- Xu, L., Bo, S., Hongde, Y. & Lei, W. (2015), ‘Evolution of rolls-royce air-cooled turbine blades and feature analysis’, *Procedia Engineering* **99**, 1482–1491.
- Yakhot, V. & Orszag, S. A. (1986), ‘Renormalization group analysis of turbulence. i. basic theory’, *Journal of scientific computing* **1**(1), 3–51.
- York, W. D. & Leylek, J. H. (2003), ‘Three-dimensional conjugate heat transfer simulation of an internally-cooled gas turbine vane’, *ASME Conference Proceedings* **2003**(36886), 351–360.
URL: <http://link.aip.org/link/abstract/ASMECP/v2003/i36886/p351/s1>
- Zhang, Q. & He, L. (2014), ‘Impact of wall temperature on turbine blade tip aerothermal performance’, *Journal of Engineering for Gas Turbines and Power* **136**(5), 052602.
- Ziugzda, J. & Zukauskas, A. (1989), ‘The influence of fluid physical parameters and of their variation on heat transfer at forced convection’, *Heat Transfer: Convective heat transfer* **1**, 453.