

Body Force Modelling for Axial and Centrifugal Compressor Pre and Post-Stall Aerodynamics

Dimitrios Lamprakis

Academic Supervisors:
Prof. Vassilios Pachidis
Dr. Mauro Righi

Industrial Supervisors:
Richard Tunstall
Steven Mazur
John Harvell

CRANFIELD UNIVERSITY

SCHOOL OF AEROSPACE, TRANSPORT AND
MANUFACTURING

PHD THESIS

ACADEMIC YEAR: 2019 - 2022

DIMITRIOS LAMPRAKIS

BODY FORCE MODELLING FOR AXIAL AND
CENTRIFUGAL COMPRESSOR PRE AND
POST-STALL AERODYNAMICS

ACADEMIC SUPERVISORS: PROF. V. PACHIDIS AND DR. MAURO
RIGHI

INDUSTRIAL SUPERVISORS: R. TUNSTALL, S. MAZUR AND J.
HARVELL

SUBMITTED TO THE SCHOOL OF AEROSPACE, TRANSPORT AND
MANUFACTURING IN PARTIAL FULFILMENT FOR THE DEGREE OF DOCTOR OF
PHILOSOPHY IN AEROSPACE ENGINEERING AT CRANFIELD UNIVERSITY.

© CRANFIELD UNIVERSITY, 2022. ALL RIGHTS RESERVED. NO PART OF THIS
PUBLICATION MAY BE REPRODUCED WITHOUT THE WRITTEN PERMISSION OF THE
COPYRIGHT OWNER.

Acknowledgments

First of all, I would like to express my gratitude to my main academic supervisor, Prof. Pachidis for offering me the opportunity to undertake such an amazing, yet challenging project within the Rolls-Royce University technology centre (UTC) at Cranfield university and for his continuous support despite his immense workload and responsibilities. I am extremely grateful and indebted to my second academic supervisor, Dr. Mauro Righi, for his continuous support throughout the project. His outstanding technical background, expertise and indispensable contribution to the project were decisive for the realization of the project.

I would also like to express my gratitude to Rolls-Royce for funding the current research project. I would like to thank in particular my industrial advisors from Rolls-Royce Indianapolis, John Harvell and Steven Mazur as well as Richard Tunstall from Rolls-Royce Bristol, for their active involvement in the project, invaluable advice, guidance and experimental data they provided.

I would like to thank my office colleagues and friends with whom we shared countless moments of joy and stress, making this long, challenging journey a more pleasant experience. In particular, I would like to thank Hannah, Hasani, Ferran, Louis, Dimitris, David John, Dario, Seyfetin and Adriano. It was a joy working and sharing the same space with you.

I would really like to thank all my friends with whom we shared many experiences, adventures and countless happy, memorable moments together. I would like to thank, with no particular order, Nico, Jesus, Ngoni, Luca, Pepo, Tom, Dario, Adriano, Hannah, Seyfetin, Pavlos, Giorgos, Christos, Alex, Sam, Priya, Eva, Fran, Dimitris, Salva, Alberto, Jordi and my girlfriend Marina.

Finally, I would like to express my perpetual gratitude to my parents, Antonios and Paraskevi, for their unconditional love and support all these years. I owe you more than I could ever possibly offer. Special thanks to my brother and housemate for the last 4 years, Ioannis, for his tremendous moral, personal and technical support. You provided me with the strength and the courage I needed during the most difficult moments of this PhD journey. *This thesis is dedicated to my family.*

“If I have seen further, it is by standing on the shoulders of Giants.” — Isaac Newton

Abstract

The operation of a jet engine is limited in a way to ensure that the manifestation, propagation and growth of local flow compressor instabilities is prevented. This inevitably leads to more conservative compressor designs with increased surge margins and reduced operating range to avoid the occurrence of unstable compressor phenomena. These instabilities are known as rotating stall and surge and their implications on the structural integrity and operability of the engine can be catastrophic.

These phenomena and the inception mechanisms that trigger their occurrence are not yet fully understood, while the knowledge gap is exacerbated when considering centrifugal or axi-centrifugal configurations. The cost of experimental campaigns for the investigation of the post-stall response of aero-engine compression systems is excessive, while the compressor is usually restricted to low rotational speeds to prevent severe structural damage. The experiments can only be performed at a more advanced, mature phase of the compressor design process. Sophisticated transient simulations using commercial CFD software offer an alternative approach but the excessive associated computational cost requirements make their real-life usage challenging. Through-flow codes combining reduced and higher-order modelling methods are a computationally efficient alternative, however very few validated implementations are reported in the literature and their capabilities are strictly limited to axial compressor configurations.

A lower-order modelling approach is developed, whereby the compression system is solved as an empty duct with body force-fields imparting turning and losses to the flow. A new body force model, applicable to all types of blades is developed. The blade curvature is fully defined in three dimensions accounting for axial, circumferential, radial forces and blade leaning. The flow-field solution is obtained transiently by solving the 2D axisymmetric Euler equations on a body force-relevant grid which ensures that the grid lines are aligned with bladed domains. The governing equations with blockage in the relative frame of reference are derived, thus replicating both metal and aerodynamic blockage effects, along with a method for the precise definition

of the additional blockage terms in convoluted ducts. The Godunov scheme coupled with 3 Riemann solvers is used to obtain the fluxes. New analytical, simplified models are derived to estimate aerodynamic blockage and mixing losses at reverse flow conditions in impeller and axial blade passages. Loss correlations are used to estimate losses at forward flow conditions, while mixed and reverse flow are treated with a separate model.

The validity and limitations of the different modelling approaches are investigated extensively using test-case scenarios and CFD data. The validation of the overall through-flow framework is carried out on axial compressor and centrifugal compressor stages using CFD and if available, experimental data. Steady-state, forward and reverse flow characteristics are in good agreement with CFD and experimental data, while the flow-field is reproduced with reasonable accuracy. Transient, post-stall simulations in centrifugal compressors are carried out and validated against experimental data. The code constitutes the first successful, validated, through-flow approach, capable of predicting the post-stall, steady-state and transient aerodynamic performance in centrifugal compressors.

Contents

Acknowledgments	v
Abstract	vii
1 Introduction	1
1.1 General context	1
1.2 Axial and centrifugal compressor performance	4
1.3 Compressor instabilities	9
1.4 Project aim and objectives	11
1.5 Contribution to knowledge	13
1.6 Thesis outline	14
2 Literature Review	17
2.1 Stall inception	18
2.2 Compressor stall	24
2.2.1 Rotating stall	24
2.2.2 Surge	28
2.3 Compressor aerodynamic performance modelling methods	36
2.3.1 0D/1D methods	37
2.3.2 2D through-flow methods	41
2.3.3 3D through-flow methods	44
2.4 Lower-order modelling methods	48
2.4.1 Body force methods	48
2.4.2 Stalled flow modelling methods	59
2.4.3 Blockage modelling methods	64
2.5 Research gap	65
2.6 Chapter summary	67

3	Methodology	71
3.1	Computational domain	72
3.1.1	Multizone manually-defined grids	74
3.1.2	Multizone adaptive grids	74
3.1.3	Blade surface grid generation	76
3.2	Solver	77
3.2.1	Governing equations in the absolute FoR	77
3.2.2	Governing equations inside stationary and rotating components	80
3.2.3	Governing equations in the relative FoR	82
3.2.4	Governing equations in ducted and bladed domains	85
3.2.5	Flux corrections at interfaces	87
3.2.6	Artificial diffusion source terms	89
3.2.7	Riemann solvers	92
3.2.8	Blockage terms evaluation	101
3.2.9	Discretisation	102
3.3	Blade metal blockage and curvature definition	106
3.3.1	Blade metal blockage and curvature interpolation	110
3.4	Body force model	112
3.4.1	Turning force model	112
3.4.2	Correction force model	116
3.4.3	Viscous force model	119
3.4.4	Distribution coefficients	123
3.4.5	Body force decomposition	126
3.5	Blockage-mixing model	129
3.5.1	Blockage model	129
3.5.2	Deviation model	132
3.5.3	Mixing loss model	135
3.6	Empirical correlations	137
3.6.1	Pressure loss coefficients and deviation angle	137
3.6.2	Impeller correlations	140
3.6.3	Vaned diffuser correlations	147
3.6.4	Inlet guide vane correlations	150
3.6.5	Axial compressor rotor/stator correlations	152
3.7	Forward, mixed and reverse flow modelling	157
3.7.1	Forward flow modelling	159
3.7.2	Reverse flow modelling	159

3.7.3	Mixed flow modelling	160
3.7.4	Shape functions	160
3.8	Plenum and throttle model	163
3.9	Chapter summary	166
4	Methods - Validity and limitations	169
4.1	Solver verification	170
4.1.1	Verification and comparison of Riemann solvers	171
4.1.2	2D axisymmetric solver verification	175
4.1.3	Blockage modelling considerations	182
4.2	Turning force model verification	186
4.3	Viscous force model verification	196
4.4	Distribution coefficient verification	201
4.5	Validation of blade curvature definition method	213
4.6	Reverse flow modelling - Validity and limitations	218
4.7	Blockage-mixing modelling implications	235
4.8	Chapter summary	243
5	Pre and post-stall aerodynamic performance	245
5.1	Forward flow validation	246
5.1.1	NASA rotor 37 - Forward flow	247
5.1.2	NASA stage 67 - Forward flow	256
5.1.3	DLR SRV2-0 - Forward flow	262
5.1.4	NASA CC3 - Forward flow	274
5.1.5	CSTAR - Forward flow	284
5.2	Reverse flow	293
5.2.1	Steady-state aerodynamic performance	294
5.2.2	Transient aerodynamic performance	308
5.3	Chapter summary	322
6	Conclusions and future work	325
6.1	Summary	325
6.2	Conclusions	328
6.3	Contribution to knowledge	332
6.4	Recommendations for future work	334
	Bibliography	337

Appendix A	355
Appendix B	363
Appendix C	367
Appendix D	379
Appendix E	385

List of Tables

4.1	Initial conditions at the left and right states of the 4 Riemann problem	
	tests.	171

List of Figures

1.1	a) Schematic of HondaJet HF 120 engine [11] and b) overview of Rolls Royce-Turbomeca RTM322 helicopter engine incorporating axi-centrifugal compressors [12].	4
1.2	a) Pressure ratio characteristics of NASA HECC centrifugal compressor with vaned diffuser [14] and b) pressure ratio and efficiency characteristic of a NASA, 10-stage axial-flow compressor [15, 16]. . .	5
1.3	a) F-16 suffering compressor stall during an airshow [18] and b) Boeing 777 suffering compressor stall during take-off [19].	6
1.4	a) Overview of the flow-field during reverse thrust operation of an installed, high bypass ratio engine at 110 knots landing speed and b) total pressure contours at different planes at the core inlet [23]. . . .	7
1.5	a) Compressor maps and transient operating lines of a non-deteriorated (solid) and deteriorated (dashed) compressor and b) overview of surge lines and surge margins between the two compressors at lower speeds [25].	8
1.6	a) Indicative compressor map including forward, reverse and rotating stall trajectory and characteristics [30] and b) Transient evolution of different rotating stall patterns during change of VSVs at the trailing edge of a stator vane [31].	10
1.7	a) Catastrophic failure of a turbocharger operating at reverse flow conditions at 135k rpm and b) Surge behaviour of the same compressor at 20k rpm [33].	11
2.1	Illustration of the circumferential propagation of a spike instability due to flow separation and change in the incidence angles of adjacent blades, based on Emmons [29].	19

2.2	a) Example of a spike disturbance in a low speed, four-stage axial-flow compressor, (original data by [46] and adapted from [47]) and b) coexistence of weak modal waves and spikes in the same compressor (original data from [39] and adapted by [47]. The compressor in the second case stalls due to spikes.	20
2.3	Stall inception criterion of Camp and Day for axial-flow compressors based on the critical rotor incidence angle and the peak efficiency point of the pressure characteristic [39, 47].	21
2.4	Pressure measurements at 105% corrected speed in the vaneless space of a centrifugal compressor with vaned diffuser a) without endwall leakage stall occurs due to spikes and b) with endwall leakage stall occurs due to modal waves [35].	22
2.5	a) Overview of compressor characteristics for the 2 different stall inception mechanisms: spikes and modal waves [35] and b) counter-rotating vortices convecting in vaneless space generating blockage [48].	23
2.6	Torque and exit duct wall pressure characteristics of a low-speed, three-stage, axial-flow compressor and overview of the rotating stall cells patterns at three flow coefficients (adapted from [51])	25
2.7	Flow inside and outside stall cell. The flow inside the stall cell is convected circumferentially through the rotors (adapted from [28]).	26
2.8	a) Pressure rise coefficient in the entire compressor, impeller and vaneless diffuser, at different diffuser inlet angles before and after mild or abrupt rotating stall [55], b) different types of impeller rotating stall and number of cells at different rotational speeds [55] and c) vaneless diffuser rotating stall experimental measurements ([54]).	27
2.9	a) Post-stall deep surge and rotating stall trajectories and the four stages of a deep-surge cycle (adapted from [58]) and b) comparison of post-stall trajectory in a high-speed axial and centrifugal compressor (adapted from [28]).	28
2.10	a) Total-to-static pressure characteristic and b) torque coefficient characteristic of the MIT 3-stage, axial-compressor rig, including rotating stall and reverse flow characteristics [59].	31

2.11 a) Deep surge cycle trajectory of a low-speed, 4-stage axial compressor for different plenum volumes and b) the same compressor undergoing rotating stall when the plenum volume is smaller than the critical one [60].	32
2.12 a) Overview of NASA CC3 compresor and position of Kulite pressure sensor and b) pressure measurements of Kulite 3 sensor of a surge event preceded by rotating stall [61, 62].	33
2.13 Velocity contours and vectors showing the flow-field in the vicinity of the NASA CC3 vaned diffuser TE (95% span) during different phases of a surge cycle [61, 62].	34
2.14 a) Schematic overview of the experimental configuration and b) experimental total-to-total pressure reverse flow characteristics of the turbocharger [63].	35
2.15 a) Pressure ratio and b) efficiency characteristics of the axi-centrifugal compressor extending beyond surge line obtained with first experimental configuration [32].	36
2.16 a) Pressure ratio and mass flow measurements during deep surge event and b) mass flow and pressure measurements at compressor inlet (1), compressor outlet (2) and downstream plenum (3) obtained with the second experimental configuration [32]).	36
2.17 Pressure ratio and efficiency predictions comparison using a) mean-line methods and 3 different loss models against experimental data (adapted from [70]) and b) mean-line methods against CFD and experimental data (adapted from [71]).	38
2.18 a) Comparison between RANS and body force simulations predictions of the pressure rise coefficient of a radial impeller and b) change in radial velocity near LE due to upstream influence introduced by the blades [85].	43
2.19 a) Experimental, CFD and body force pressure ratio data of a centrifugal compressor with vaned diffuser and b) backwards-travelling modal waves obtained with body force simulations [49].	45
2.20 Experimental and through-flow predictions of entire compressor map, of the 4-stage C106 rig by a) Longley [91] and b) Righi [27].	46
2.21 Axial Mach number contours showing the evolution of rotating stall cell patterns at different rotors and shaft revolutions, predicted by ACRoSS and uRANS simulations [58].	47

2.22	Illustration of the smeared camberline representation of the IBMSG model [96]).	49
2.23	a) Benneke's surface model of tangential body force component as a function of relative Mach number and relative flow angle and b) data corresponding to 3 speedlines, collapsing into a single curve, compromising the accuracy of the response surface model [49].	51
2.24	Representation of Gong's blade passage model and body force components (adapted from [99])	53
2.25	Overview of a) Brand's body force model [88] and b) Kottapalli's body force model [85].	56
2.26	Schematic overview of Longley's simplified reverse flow model (adapted from [91]).	60
3.1	a) Adaptive and b) hybrid mesh approach of SRV2-0 centrifugal compressor [107] with inlet bell mouth and vaneless diffuser generated by synthesizing ducted, bladed and user-defined subdomains.	75
3.2	Adaptive mesh approach of SRV2-0 centrifugal compressor [107] with the addition of variable node distribution transition regions.	76
3.3	Flux correction at the interface adjoining stationary and rotating domains.	89
3.4	Wave structure of the solution of the exact Riemann problem of the x-split, 3D Euler equations (adapted from [108])	93
3.5	Wave structure of a) HLL approximate Riemann solver and b) HLLC Riemann problem (adapted from [108]).	94
3.6	Index nomenclature along computational grid.	103
3.7	a) Camber surface grid and meridional lines passing through the nodes of the grid for the estimation of blade curvature. Definition of the b) blade and lean angle and c) flow-path angle (defined in the meridional plane).	107
3.8	a) Pressure, suction and camber surface mesh of an axial rotor blade and b) definition of blockage parameter.	109
3.9	Transformation of arbitrary interpolating quadrilateral to a unit square quadrilateral.	110
3.10	Body force balance along a relative streamline for the derivation of the body force model.	113

3.11	Overview of correction force model velocity components due to flow misalignment with respect to the imposed relative flow path.	117
3.12	Viscous force acting parallel to the relative streamlines generating entropy and stagnation pressure loss.	120
3.13	Extraction and synthesis process of viscous force distribution coefficients using averaged streamwise and spanwise distribution profiles extracted from CFD simulations.	125
3.14	Schematic overview of the simplified axial blade-duct configuration showing the assumptions of the blockage-mixing model. Mixing occurs in an entirely axial, constant area duct downstream the blade LE. . .	130
3.15	Schematic overview of the simplified radial blade-duct configuration showing the assumptions of the blockage-mixing model.	133
3.16	Definition of angles and impeller nomenclature in accordance to [68] .	140
3.17	Definition of angles and blade nomenclature in accordance to [64] . . .	153
3.18	Imposed flow angle and deviation at different flow regimes.	158
3.19	Modified curvature distributions extending 10% 30% and 60% in the streamwise direction from the TE. The flow is entering from the TE (reverse flow).	161
4.1	Comparison between analytical solution and different Riemann solvers of the first Sod shock tube problem at time $t = 0.25$ units.	172
4.2	Comparison between analytical solution and different Riemann solvers of the second Sod shock tube problem at time $t = 0.15$ units.	173
4.3	Comparison between analytical solution and different Riemann solvers of the third Sod shock tube problem at time $t = 0.012$ units.	174
4.4	Comparison between analytical solution and different Riemann solvers of the forth Sod shock tube problem at time $t = 0.035$ units.	175
4.5	Computational domain of centrifugal compressor SRV2-0 duct generated by a) the code and b) ANSYS FLUENT mesher.	176
4.6	a)Pressure, b) axial velocity, c) radial velocity and d) Mach number contours along the compressor SRV2-0 duct, obtained from FLUENT.	177
4.7	a)Pressure, b) axial velocity, c) radial velocity and d) Mach number contours along the compressor SRV2-0 duct, derived from the custom solver.	178

4.8	Distribution of Impeller SRV2 a) metal blockage, b) radial blockage gradient and c) axial blockage gradient. The presence of large blockage gradients at regions upstream and downstream the LE and TE respectively are highlighted in the zoomed regions.	179
4.9	Impeller SRV2 duct with metal blockage, rotating and stationary domains and no body forces to the verify flux correction and blockage gradient evaluation methods.	180
4.10	Mach number, stagnation pressure and temperature contours obtained from the custom solvers in the relative and absolute FoR.	182
4.11	Blockage smoothening at the LE and TE of the NASA CC3 blades to reduce numerically induced pressure losses.	183
4.12	Reduction in mesh-related errors for different levels of blockage smoothening (aerodynamic) in CSTAR duct and overview of the blockage and total pressure contours with and without smoothening.	185
4.13	Blade, imposed and actual relative flow angle distributions at the hub, midspan and shroud in impeller SRV2 passage.	187
4.14	Base turning, correction and turning force contours in impeller SRV2 passage.	188
4.15	Blade, imposed and actual absolute flow angle distributions at the hub, midspan and shroud in NASA CC3 vaned diffuser passage.	189
4.16	Base turning, correction and turning force contours in NASA CC3 vaned diffuser passage.	190
4.17	Blade, imposed and actual relative flow angle distributions at the hub, midspan and shroud in rotor 67 passage.	191
4.18	Base turning, correction and turning force contours in rotor 67 passage.	192
4.19	Blade, imposed and actual absolute flow angle distributions at the hub, midspan and shroud along stator 67 passage.	193
4.20	Blade, imposed and actual relative flow angle distributions at the hub, midspan and shroud in rotor 37 passage.	194
4.21	Base turning, correction and turning force contours in rotor 37 passage.	195
4.22	Isentropic pressure and efficiency characteristics of a) Rotor 37 and b) impeller SRV2.	196
4.23	Rotor 37 a) pressure and efficiency characteristics at 60% speed obtained by varying the imposed losses, b) comparison of imposed and actual losses, c) viscous force coefficient and d) viscous force at operating point (1).	197

4.24 Impeller SRV2 modelled as stator a) pressure and efficiency characteristics obtained by varying the imposed losses, b) comparison of imposed and actual losses, c) viscous force coefficient and d) viscous force at point at point (1).	199
4.25 Impeller SRV2 rotating at 60% NRT a) comparison of imposed and actual losses, c) viscous force coefficient and d) viscous force at point at point (1).	200
4.26 Rotor 37 a) pressure and b) efficiency characteristics obtained by varying the imposed deviation angle using uniform and CFD deviation coefficients and no losses.	202
4.27 a) Total pressure and b) total temperature ratio profile at the TE of rotor 37 obtained using uniform and CFD viscous force coefficients. The distributions of the deviation angle corresponding to operating points (1) and (2) are shown in c) and d).	203
4.28 Comparison of imposed and measured flow angles at the TE using different deviation coefficients.	204
4.29 Rotor 37 a) pressure and b) efficiency characteristics obtained by varying the imposed total pressure loss and applying a zero deviation angle using uniform and CFD deviation coefficients.	205
4.30 a) Total pressure and b) total temperature ratio profile at the TE of rotor 37 obtained using uniform and CFD viscous force coefficients. The distributions of the viscous force corresponding to operating points (1) and (2) are shown in c) and d).	206
4.31 Comparison of imposed and measured pressure losses at the TE using different viscous force coefficients.	207
4.32 Pressure ratio and efficiency characteristics of rotor 37 obtained using different combinations of deviation and viscous force distribution coefficients at 60% design speed.	208
4.33 Spanwise profiles of pressure ratio, temperature ratio and efficiency of rotor 37 at near choke, off-design and near stall operating conditions.	209
4.34 Pressure ratio and efficiency characteristics of CSTAR centrifugal compressor obtained using different viscous force distribution coefficients.	211
4.35 Spanwise profiles of pressure ratio, temperature ratio and efficiency of CSTAR centrifugal compressor at near choke, off-design and near stall operating conditions.	212

4.36	Meridional projection of Rotor 37 grid a) used by [85] and b) generated by the code.	214
4.37	Blade angle, flow path angle, lean angle and blockage parameter distributions of Rotor 37 computed by [85].	215
4.38	Blade angle, flow path angle, lean angle and blockage parameter distributions of Rotor 37 computed by the code.	216
4.39	a) Meridional projection of impeller SRV2 grid generated by the code and experimental and computed streamwise distributions of blade angles at b) the hub and c) the shroud.	217
4.40	a) Deviation angle and b) loss coefficient predictions of the analytical blockage-mixing model with respect to the aerodynamic blockage due to constant area flow mixing.	219
4.41	Variable area mixing a) Deviation angle against radius ratio for different blockage parameters b) deviation angle against blockage for different radius ratios, c) mixing losses against radius ratio for different blockage parameters d) mixing losses against blockage for different radius ratios.	220
4.42	Comparison of rotor 37 aerodynamic blockage predictions at reverse flow conditions using the new, Longley's and Powers et al. model. . .	222
4.43	Flow-field and comparison of impeller CC3 aerodynamic blockage predictions at reverse flow conditions using the new, Longley's and Powers et al. model.	223
4.44	Flow-field and comparison of impeller SRV2 aerodynamic blockage predictions at reverse flow conditions using the new, Longley's and Powers et al. model.	223
4.45	Flow-field and comparison of NASA CC3 vaned diffuser aerodynamic blockage predictions at reverse flow conditions using the new, Longley's and Powers et al. model.	224
4.46	a) Area-averaged deviation angle at different streamwise positions in NASA CC3 impeller and b) deviation angle contour at point (1), corresponding to a high pressure ratio setting.	226
4.47	a) Area-averaged deviation angle at different streamwise positions in SRV2 impeller and b) deviation angle contour at point (1), corresponding to a low pressure ratio setting.	226

4.48 a) Area-averaged deviation angle at different streamwise positions in NASA CC3 vaned diffuser, b) deviation angle contour and c) absolute velocity streamlines at point (1), corresponding to a high inlet dynamic head.	227
4.49 a) Area-averaged deviation angle at different streamwise positions in CSTAR vaned diffuser, b) deviation angle contour and c) absolute velocity streamlines at point (1), corresponding to a high inlet dynamic head.	228
4.50 a) Area-averaged deviation angle at different streamwise positions in rotor 37 passage and b) deviation angle contour at point (1), corresponding to a high pressure ratio setting.	229
4.51 Comparison of streamwise rotary stagnation pressure loss distribution in impeller SRV2 passage at reverse flow conditions and 60% speed obtained a) from the code and b) from CFD simulations.	231
4.52 Comparison of streamwise rotary stagnation pressure loss distribution in impeller CC3 passage at reverse flow conditions and 80% speed obtained a) from the code and b) from CFD simulations.	231
4.53 Comparison of rotary stagnation pressure loss distribution in the streamwise direction in vaned diffuser CC3 at reverse flow conditions and 60% impeller speed obtained a) from the code and b) from CFD simulations.	232
4.54 Comparison of rotary stagnation pressure loss distribution in the streamwise direction in vaned diffuser CC3 at reverse flow conditions and 80% impeller speed obtained a) from the code and b) from CFD simulations.	233
4.55 Mixing losses obtained by CFD simulations compared by predictions of the analytical model a) at the inlet duct and b) at the vaneless space of NASA CC3 compressor at reverse flow conditions.	234
4.56 Comparison of a) total temperature ratio, b) total pressure ratio and c) flow angle spanwise distribution at the outlet obtained by CFD simulations and the code using CFD and uniform coefficients.	236
4.57 Effect of deviation due to mixing-out of the flow on the 60% reverse flow pressure and torque characteristics of impeller SRV2. Characteristics with and without deviation are compared at 20% and 40% aerodynamic blockage.	238

4.58	Effect of passage and additional mixing losses on the 80% reverse flow pressure and torque characteristics of impeller SRV2 with 20% aerodynamic blockage. The position of the characteristic due to the addition of losses and deviation is shown relative to the ideal one. . .	239
4.59	a) Imposed flow angles corresponding to different modified curvature lengths and streamwise distribution of mass flow-averaged rotary stagnation pressure losses in impeller SRV2 passage for b) 10% c) 30% and d) 60% modified curvature.	241
4.60	Effect of modified curvature length on the 40% reverse flow pressure and torque characteristics of impeller SRV2. aerodynamic blockage, mixing losses and deviation are neglected.	242
5.1	3D visualization of axial compressor NASA rotor 37. The main computational duct grid along with the camber surface grid is also shown.	247
5.2	Overview of rotor 37 computational domain used for single-passage RANS simulations.	248
5.3	Overview of rotor 37 a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.	249
5.4	Adaptive computational domain of rotor 37 blade generated by the code to obtain the flow-field solution.	249
5.5	Comparison of NASA rotor 37 experimental pressure ratio and efficiency characteristics against CFD and code predictions using empirical models coefficients proposed by Aungier.	250
5.6	Comparison of NASA rotor 37 experimental pressure ratio and efficiency characteristics against CFD and code predictions using different set of empirical models coefficients.	251
5.7	Efficiency, total pressure and temperature ratio spanwise profiles downstream the TE of rotor 37 obtained by through-flow and CFD simulations at three different operating points on the 60% characteristic. . .	252
5.8	Comparison of spanwise flow angle distributions obtained by through-flow and CFD simulations at 3 operating points on the 60% characteristic of rotor 37.	253
5.9	Flow angle contours at a high mass flow operating point at 60% speed obtained from a) the code and b) CFD simulations.	254

5.10	Comparison of through-flow against CFD total pressure and temperature contours of rotor 37, at a high mass-flow operating point on the 60% characteristic.	255
5.11	Comparison of through-flow against CFD meridional Mach number and static pressure contours of rotor 37, at a high mass-flow operating point on the 60% characteristic.	256
5.12	3D visualization of axial fan stage NASA 67. The main computational duct grid along with the camber surface grid is also shown.	257
5.13	Overview of rotor 67 a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.	257
5.14	Overview of stator 67 a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.	258
5.15	Adaptive computational domain of stage 67 blade generated by the code to obtain the flow-field solution.	258
5.16	Blade angles and metal blockage parameter contours of rotor 67 blade.	259
5.17	Blade angles and metal blockage parameter contours of stator 67 blade.	260
5.18	Comparison of NASA stage 67 experimental pressure ratio and efficiency characteristics against code predictions using empirical models coefficients proposed by Aungier.	261
5.19	Comparison of NASA stage 67 experimental pressure ratio and efficiency characteristics against code predictions using different set of coefficients for the empirical models.	262
5.20	3D visualization of SRV2 centrifugal compressor with vaneless diffuser. The main computational duct grid along with camber surface grid is also shown.	263
5.21	Overview of SRV2-0 compressor computational domain with vaneless diffuser used for single-passage RANS simulations.	264
5.22	Overview of impeller SRV2 a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.	265
5.23	Adaptive computational domain of impeller SRV2 generated by the code to obtain the flow-field solution.	265
5.24	Blade angles and metal blockage parameter contours of SRV2 impeller.	266

5.25 Comparison of pressure ratio and efficiency characteristics of SRV2-0 compressor with vaneless diffuser obtained by CFD and through-flow simulations.	267
5.26 Comparison of total pressure ratio (top), total temperature (middle) and efficiency (bottom) profiles obtained by CFD and through-flow simulations of SRV2-0 compressor at three different operating conditions along the 60% design speed.	269
5.27 Spanwise distribution of relative flow angle at the TE of SRV2-0 impeller at different operating conditions at 60% design speed.	270
5.28 Efficiency, total pressure and temperature ratio spanwise distributions at the TE, middle of the vaneless diffuser and outlet of the SRV2-0 compressor at operating point (2) of the 60% design speed characteristic.	271
5.29 Comparison of total pressure and temperature contours of SRV2 impeller at operating point (2) of the 60% design speed characteristic.	272
5.30 Comparison of static pressure and meridional Mach number contours of SRV2 impeller at operating point (2) of the 60% design speed characteristic.	273
5.31 3D visualization of centrifugal compressor NASA CC3 a) with vaneless and b) without vaned diffuser. The main computational duct grid along with the camber surface grid is also shown.	274
5.32 Overview of NASA CC3 impeller and vaned diffuser computational domains used for single-passage RANS simulations.	275
5.33 Overview of NASA CC3 impeller a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.	276
5.34 Overview of NASA CC3 vaned diffuser a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.	276
5.35 Computational domain of NASA CC3 compressor generated by the code using the adaptive mesh approach.	277
5.36 Blade angles and metal blockage parameter contours of NASA CC3 impeller blade.	278
5.37 Blade angles and metal blockage parameter contours of NASA CC3 vaned diffuser blade.	279

5.38 Comparison of pressure ratio and efficiency characteristics of NASA CC3 compressor with vaneless diffuser obtained by CFD and through-flow simulations.	280
5.39 Comparison of pressure ratio and efficiency characteristics of NASA CC3 compressor with vaned diffuser obtained by CFD and through-flow simulations.	281
5.40 Comparison of NASA CC3 compressor (vaned diffuser) stagnation pressure and temperature contours obtained by the through-flow code and the CFD model.	282
5.41 Comparison of NASA CC3 compressor (vaned diffuser) static pressure and meridional Mach number contours obtained by the through-flow code and the CFD model.	283
5.42 3D visualization of CSTAR centrifugal compressor. The main computational duct grid along with the camber surface grid is also shown. .	284
5.43 Overview of CSTAR impeller and vaned diffuser computational domains used for single-passage RANS simulations.	285
5.44 Overview of CSTAR impeller a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.	286
5.45 Overview of CSTAR vaned diffuser a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.	286
5.46 Computational domain of CSTAR compressor generated by the code using the adaptive mesh approach.	287
5.47 Comparison of pressure ratio and efficiency characteristics of CSTAR compressor obtained by CFD and through-flow simulations.	288
5.48 Comparison of total pressure ratio (top), total temperature (middle) and efficiency (bottom) profiles obtained by CFD and through-flow simulations of CSTAR compressor at three different operating conditions and 80% design speed.	289
5.49 Comparison of absolute flow angle spanwise distribution in the vaneless space of CSTAR compressor.	290
5.50 Variations of total temperature ratio profile at different locations downstream the vaned diffuser obtained by CFD simulations and the code using CFD and uniform coefficients.	291

5.51 Comparison of a) total temperature ratio, b) total pressure ratio and c) flow angle spanwise distribution at the outlet obtained by CFD simulations and the code using CFD and uniform coefficients.	291
5.52 Comparison of CSTAR compressor stagnation pressure and temperature contours obtained by the through-flow code and the CFD model.	292
5.53 Comparison of CSTAR compressor static pressure and meridional Mach number contours obtained by the through-flow code and the CFD model.	293
5.54 Comparison of a) total-to-total pressure ratio and b) total-to-static pressure ratio c) torque and d) torque parameter reverse flow characteristics of compressor SRV2-0.	296
5.55 Three-dimensional streamlines in SRV2-0 compressor, illustrating the recirculation regions at the inlet domain and near the TE obtained from steady-state, single-passage RANS simulations.	297
5.56 Relative Mach number contours and velocity vectors along SRV2-0 impeller passage obtained from CFD simulations, showing regions of separated flow, recirculation and shock waves during reverse flow operation near the a) hub, b) midspan and c) shroud.	298
5.57 Static pressure and streamwise velocity contours comparison of SRV2-0 compressor with vaneless diffuser obtained by the through-flow code and the CFD model at steady-state reverse flow conditions and 40% speed.	299
5.58 Comparison of a) total-to-total pressure ratio and b) total-to-static pressure ratio c) torque and d) torque parameter reverse flow characteristics of compressor CC3 with vaneless diffuser.	300
5.59 Three-dimensional streamlines in NASA CC3 compressor passage with vaneless diffuser, illustrating the recirculation regions at the inlet domain and near the TE obtained from steady-state, single-passage RANS simulations.	301
5.60 Relative Mach number contours and velocity vectors along impeller passage of NASA CC3 compressor with vaneless diffuser obtained from CFD simulations, showing regions of separated flow, recirculation and shock waves during reverse flow operation near the a) hub, b) midspan and c) shroud.	302

5.61	Static pressure and streamwise velocity contours comparison of CC3 compressor with vaneless diffuser obtained by the through-flow code and the CFD model at steady-state reverse flow conditions and 80% speed.	303
5.62	Comparison of a) total-to-total pressure ratio and b) total-to-static pressure ratio c) torque and d) torque parameter reverse flow characteristics of compressor CC3 with vaned diffuser.	304
5.63	Three-dimensional streamlines in NASA CC3 compressor with vaned diffuser, illustrating the recirculation regions at the inlet domain and near the TE obtained from steady-state, single-passage RANS simulations.	305
5.64	Absolute Mach number contours and velocity vectors along impeller passage of NASA CC3 compressor with vaneless diffuser obtained from CFD simulations, showing regions of separated flow, recirculation and shock waves during reverse flow operation near the a) hub, b) midspan and c) shroud.	306
5.65	Static pressure and streamwise velocity contours comparison of CC3 compressor with vaned diffuser obtained by the through-flow code and the CFD model at steady-state reverse flow conditions and 60% speed.	307
5.66	Pressure ratio versus mass flow during deep surge of NASA CC3 compressor with vaneless diffuser with a plenum volume of $1m^3$	309
5.67	Torque versus mass flow during deep surge of NASA CC3 compressor with vaneless diffuser and a plenum volume of $1m^3$	310
5.68	Streamwise velocity contours at different operating conditions during a deep surge cycle of NASA CC3 compressor with vaneless diffuser and a plenum volume of $1m^3$	311
5.69	Pressure ratio versus mass flow during deep surge cycle of NASA CC3 compressor with vaneless diffuser and different plenum volumes: a) $3m^3$, b) $5m^3$ c) $7.5m^3$ d) comparison of surge trajectories.	312
5.70	Pressure ratio versus mass flow during deep surge of NASA CC3 compressor with vaned diffuser and a plenum volume of $1m^3$	314
5.71	Torque versus mass flow during deep surge of NASA CC3 compressor with vaned diffuser with a plenum volume of $1m^3$	315
5.72	Streamwise velocity contours at different operating conditions during a deep surge cycle of NASA CC3 compressor with vaned diffuser with a plenum volume of $1m^3$	316

5.73	Pressure ratio versus mass flow during deep surge cycle of NASA CC3 compressor with vaned diffuser and different plenum volumes: a) $3m^3$, b) $5m^3$ c) $7.5m^3$ d) comparison of surge trajectories.	317
5.74	NASA CC3 experimental pressure recordings during two different surge events at a) vaned diffuser throat and b) vaned diffuser throat and impeller inlet (adapted from [61, 62]).	318
5.75	NASA CC3 through-flow pressure recordings at vaned diffuser throat and impeller inlet during surge events obtained with a plenum volume of a) $0.25m^3$ and b) $0.15m^3$	319
5.76	Comparison of NASA CC3 through-flow pressure recordings at vaned diffuser throat against experimental measurements during surge a surge event. Three consecutive cycles corresponding to the surge event are shown.	320
5.77	a) Pressure ratio and b) torque versus mass flow of the simulated experimental surge event of NASA CC3 compressor with vaned diffuser and a plenum volume of $0.25m^3$	321
5.78	Comparison of experimental and simulated pressure traces at vaned diffuser throat during a surge cycle with plenum volume of a) $0.19m^3$ and b) $0.25m^3$	322

Nomenclature

Latin characters

a	Speed of sound $[m/s]$
a_0	Speed of sound using stagnation properties $[m/s]$
$\tilde{a}$	Wave strength
A	Area $[m^2]$
a/c	Position of maximum camberline $[-]$
b	Blockage parameter $[-]$
B	Greitzer parameter / Tip area blockage $[-]/[-]$
$\mathbf{B}$	Blockage gradient vector
c	Blade chord $[-]$
C	Distribution coefficient/ Generic coefficient $[-]/[-]$
c_f	Skin friction coefficient $[-]$
c_{fl}	Laminar skin friction coefficient $[-]$
c_{ft}	Turbulent skin friction coefficient $[-]$
c_v	Specific heat at constant volume $[J/(kg\ K)]$
c_p	Specific heat at constant pressure $[J/(kg\ K)]$
$\mathbf{CV}$	Conserved variable vector
D	Diameter $[m]$
D_{eq}	Equivalent diffusion factor $[-]$
d_H	Hydraulic diameter $[m]$
e	Specific internal energy $[J/kg]$
E_0	Specific total energy $[J/kg]$
E_r	Specific relative total energy $[J/kg]$
f	Skin friction coefficient $[-]$
$\mathbf{f}$	Body force vector $[N/m^3]$
$\mathbf{f}_\tau$	Force vector due to shear stresses $[N/m^3]$
$\mathbf{F}$	Streamwise flux vector
$\mathbf{G}$	Spanwise flux vector
h	Static enthalpy / Channel height $[J/kg]/[m]$
H_0	Stagnation enthalpy $[J/kg]$
I	Rothalpy $[J/kg]$
i	Streamwise index / incidence angle $[-]/[rad]$
i_{PS}	Positive stall incidence angle $[rad]$

i_{NS}	Negative stall incidence angle [rad]
I_B	Input work coefficient [—]
j	Spanwise index
$\mathbf{K}$	Eigenvector
$\hat{\mathbf{k}}$	Unit vector normal to $\hat{\mathbf{l}}$ and $\hat{\mathbf{p}}$
$\hat{\mathbf{l}}$	Unit vector parallel to the relative streamline
L_B	Length of blade mean camberline [m]
L_m	Length of blade mean meridional streamline [m]
L_{SB}	Length of splitter blade mean camberline [m]
Ma	Mach number [—]
m	Meridional coordinate [m]
$\hat{\mathbf{m}}$	Meridional unit vector
m_f	Coordinate parallel to streamwise cell faces [m]
n_f	Coordinate perpendicular to m_f and θ axes [m]
n	Spanwise coordinate, perpendicular to m and θ axes
$\hat{\mathbf{n}}$	Unit vector normal to cell face area
N_{ef}	Effective number of blades [—]
P	Static pressure [Pa]
P_0	Stagnation pressure [Pa]
PV	Velocity pressure [Pa m/s]
R	Gas constant/Angle stall margin [J/kg K]/[rad]
R_{curv}	Local radius of curvature [m]
Re	Reynolds number [—]
$\mathbf{r}$	Radial position vector [m]
r	Radius [m]
s	Entropy / Blade pitch [J/(K kg)]/[m]
$\hat{\mathbf{s}}$	Unit vector parallel to nodes in the spanwise direction
S	Wave speed [m/s]
t	time / thickness [s]/[m]
$\hat{\mathbf{t}}$	Unit vector parallel to turning force
T	Static temperature [K]
T_0	Stagnation temperature [K]
$\mathbf{u} = [u_x, u_\theta, u_r]$	Absolute velocity vector [m/s]
$u_{m,f}$	Velocity component normal to the cell face [m/s]
$u_{n,f}$	Velocity component parallel to the cell face [m/s]
$\mathbf{u}_{rot} = \Omega \times \mathbf{r}$	Blade speed vector [m/s]

$\mathbf{U}$	Primitive variable vector
V	Absolute velocity magnitude/Volume [m/s]/[m ³]
V_{mer}	Meridional velocity [m/s]
$\mathbf{w} = [w_x, w_\theta, w_r]$	Relative velocity vector [m/s]
W_{ex}	Power exchange term due to body forces [W/m ³]
W_τ	Power exchange term due to shear stresses [W/m ³]
W	Relative velocity magnitude [m/s]
X	Choking trigger [—]

Greek characters

α	Angle of attack [rad]
β	Relative flow angle [rad]
γ	Specific heat ratio / Stagger angle [—]/[rad]
δ	Deviation angle [rad]
ΔP_0	Average stagnation pressure loss [Pa]
θ	Circumferential position / camber angle [rad]/[rad]
$\theta_{f,m}$	Angle of streamwise cell faces [rad]
κ	Blade metal angle [rad]
$\hat{\mathbf{k}}$	Unit vector parallel to camberline
λ	Lean angle / Eigenvalue / Tip distortion factor [rad]/[—]/[—]
Λ	Generic quantity
$\mathbf{\Lambda}$	Generic differentiable vector field
μ^*	Dynamic viscosity [Pa s]
ν^*	Artificial kinematic viscosity [m ² /s]
$\hat{\mathbf{v}}$	Unit vector parallel to the turning force
ξ	Normalised spanwise coordinate [—]
ξ_{tr}	Normalised transition point coordinate [—]
ρ	Density [kg/m ³]
σ	Slip factor / solidity [—]/[—]
τ	Shear stress/Torque [Pa]/[Nm]
$\phi_{f,m}$	is the cell face vector angle with respect to the x-axis [rad]
ϕ	Flow path angle [rad]
Φ	Flow coefficient [—]
ψ	Normalised spanwise coordinate [—]
$\bar{\omega}$	Pressure loss coefficient [—]

$\Omega = [\Omega_x, \Omega_\theta, \Omega_r]$ Angular velocity vector [rad/s]

Subscripts

-1	IGV inlet
0	Stagnation property / IGV outlet
1	Rotor inlet
2	Rotor outlet
3	Diffuser inlet
4	Diffuser outlet
<i>act</i>	Actual/measured quantity
<i>abs</i>	Quantity in the absolute frame of reference
<i>atm</i>	Atmospheric property
<i>bl</i>	Related to blade
<i>BL</i>	Boundary layer property
<i>ch</i>	Choking property
<i>CL</i>	Related to clearance loss
<i>CS</i>	Camber surface property
<i>cor</i>	Related to correction force / Corrected property
<i>cyl</i>	Related to cylindrical coordinates
<i>dev</i>	Deviation angle quantity
<i>fb</i>	Full blade property
<i>h</i>	Quantity at the hub
<i>id</i>	Ideal property
<i>imp</i>	Imposed property / impeller property
<i>mix</i>	Mixing property
<i>m</i>	Meridional property
<i>NS</i>	Negative stall property
<i>K</i>	Left or right state
<i>L</i>	Left state
<i>LE</i>	Quantity at the leading edge
<i>PCHIP</i>	Piecewise cubic Hermite interpolating polynomial
<i>PS</i>	Pressure surface property / Positive stall property
<i>PL</i>	Plenum volume property
<i>R</i>	Right state

<i>ref</i>	Reference quantity
<i>rel</i>	Quantity in the relative frame of reference
<i>rot</i>	Related to Coriolis and centripetal forces
<i>sb</i>	Splitter blade property
<i>span</i>	Quantity in the spanwise direction
<i>SEP</i>	Separated flow property
<i>SS</i>	Suction surface property
<i>SF</i>	Related to skin friction
<i>stream</i>	Quantity in the streamwise direction
<i>t</i>	Quantity at the tip
<i>T</i>	Throttle property
<i>TE</i>	Quantity at the trailing edge
<i>th</i>	Quantity at the throat
<i>turn</i>	Turning force quantity
<i>visc</i>	Viscous force quantity
τ	Related to shear stresses
Ω	Related to angular velocity or blade speed

Superscript

- * Property in the star region / artificial property / modified term / minimum loss property
n Current timestep index

Acronyms

ACRoSS	Axial compressor rotating stall and surge code
AIRS	Abrupt impeller rotating stall
ANRS	Adaptive noniterative Riemann solver
BC	Boundary condition
CFD	Computational fluid dynamics
CFL	Courant-Friedrich-Lewi's number
DPIV	Digital particle imaging velocimetry
EEC	Electronic engine control
FADEC	Full authority digital engine control
FEGV	Fan exit guide vane

FoR	Frame of reference
FVM	Finite volume method
HECC	High efficiency centrifugal compressor
HLL	Harten-Lax-van-Leer
HLLC	Harten-Lax-van-Leer-Contact
HPC	High-pressure compressor
ICAO	International civil aviation organization
IGV	Inlet guide vane
IPC	Intermediate-pressure compressor
LE	Leading edge
LHS	Left hand side
MIRS	Mild impeller rotating stall
MUSCL	Monotonic upwind scheme for conservation laws
NASA	National aeronautics and space administration
OPR	Overall pressure ratio
PIRS	Progressive impeller rotating stall
PVRS	Primitive variables Riemann solver
RANS	Reynolds-averaged Navier-Stokes
RHS	Right hand side
RPM	Revolutions per minute
SLC	Streamline curvature
SST	Shear stress transport
TET	Turbine entry temperature
TRRS	Two-rarefaction Riemann solver
TVD	Total variation diminishing
VSV	Variable stator vane

Chapter 1

Introduction

This chapter provides the general context of the present work, analysing the trends in the aviation sector, the path towards de-carbonisation and the design strategy of aero-engine manufacturers to meet the ever-increasing stringent emission requirements. The characteristics and differences of axial and centrifugal compressors during stable operation are briefly discussed. Phenomena leading to the onset of full-lengthscale instabilities in gas turbine compression systems are subsequently analysed. Subsequently, introductory concepts on compressor post-stall behaviour, the associated phenomena and their implication on the engine operability is provided. The chapter is concluded by listing the aims and objectives of the project, analysing the contribution to knowledge and describing the thesis outline.

Contents

1.1	General context	1
1.2	Axial and centrifugal compressor performance	4
1.3	Compressor instabilities	9
1.4	Project aim and objectives	11
1.5	Contribution to knowledge	13
1.6	Thesis outline	14

1.1 General context

The data published by the international civil aviation organization(ICAO) indicate a dramatic rise in civil air transport demand over the last 50 years, with the annual global number of passengers increasing from roughly 0.31 billion in 1970, to almost

4.56 billion in 2019 [1]. The implementation of novel, more efficient technologies over the years resulted in substantial reduction in the manufacturing and operational cost of air transportation through the implementation of novel, more efficient technologies, which in turn, drastically reduced the transportation cost per passenger. The demand in air-transport was further amplified due to additional cost reductions achieved via air transport agreements and low-cost airlines. This further contributed to the reduction of air transportation costs and further enhanced the rising trend in civil aviation. The flourishing of the air transport market lead to a rapid growth of the aerospace industry and consequently the civil aircraft industry.

The rapid growth of the aviation industry, detrimentally impacted the environment since it drastically increased the emissions of environmentally unfriendly particles, which comprise carbon dioxide, nitrogen oxides, hydrocarbons, soot and other particles [2]. One of the primary concerns of the global community was the limitation of the environmental footprint induced by the aviation industry [3]. The Advisory Council for Aeronautical Research in Europe was established in 2001, aiming to regulate aviation gas and noise emissions within the European Union. These ambitious targets were reflected on the objectives defined by the VISION 2020 plan [4], which included a reduction by 50% and 80% in CO₂ and NO_x emissions respectively, as well as the reduction by 50% in the perceived external noise. The updated roadmap of ACARE, for noise and emission reductions in civil aviation until 2050, was published within the framework of Flightpath 2050 [5]. The ambitious goals include reductions of 75% and 90% in CO₂ and NO_x emissions respectively along with a 65% reduction in noise emissions with respect to year 2000 technology level aircraft.

Hydrogen has long been envisaged as a potential replacement fuel for kerosene due its prospect to provide rapid de-carbonisation of the aviation industry and is currently a very active research-field. Nevertheless, its low density and the low temperature needed to maintain its liquid form, requires large fuel tanks which translate to substantial changes to the airframe design, advanced heat management solutions, while its potential to constitute an alternative to conventional fuels is seemingly feasible strictly in long-range aircraft [6]. Despite the significant research efforts and technology advancements, the technology is not expected to achieve the intended maturity level soon. Instead, the next generation of civil turbofan engines is geared towards higer BPR, lower fan pressure ratios (FPR) and lower specific thrust designs, in an attempt to reduce the engine's specific fuel consumption (SFC) and increase propulsive efficiency [7]. Further improvements are potentially feasible by increasing the overall pressure ratio (OPR) and turbine entry temperature (TET)

which in turn would result in higher overall thermodynamic cycle efficiency [8]. These improvements are inevitably associated with increased thermo-mechanical loads and fatigue and are therefore limited to current material and blade cooling technologies [9]. A design aspect associated with the increasingly higher mass flow and pressure ratio compressor designs is the stable operation of the compression system. During unstable compressor operation, the large mass flow and the high energy imparted into the flow exert large mechanical loads on all engine components. The ability of the engine to recover from such events and contain debris in case of catastrophic failure is a certification requirement and becomes more challenging as OPR increases.

Axial and centrifugal compressors constitute one of the most essential components of gas turbine engines and are extensively used by the industry in a wide range of land-based (e.g. energy production), automotive, marine and aerospace applications. In the civil aerospace sector, the ever-increasing efforts for higher propulsive efficiencies by means of larger mass-flow capacities, can only be accommodated by axial-flow compressors. The incorporation of centrifugal compressors in larger civil, aero-engines is practically not feasible due to the incapability of centrifugal compressors to accommodate the large amounts of mass-flow passing through the engine core without this resulting to an excessively oversized impeller, especially in the radial direction. This would inevitably contradict the design efforts for smaller engine cores and larger bypass duct designs. Centrifugal compressors are extensively used in the automotive industry as well as in turbo-shaft and small turbofan engines, where the mass-flow capacity requirements are substantially lower due to their ability of generating much higher pressure ratios compared to axial-flow compressors. Indicatively, a pressure ratio of more than 6:1 can be achieved by a single stage centrifugal compressor compared to approximately 2:1 of a single-stage axial stage. This is achieved due to the change in the mean streamline radius which imparts additional energy to the flow due to increase in the rotational velocity as the flow moves from the LE to the TE. While centrifugal compressors provide significant benefits in terms of pressure rise, they generally are less efficient than axial-flow compressors [10]. In the automotive industry, emissions reduction regulations drove the engine downsizing trend which was accompanied by extensive application of turbochargers with high-pressure ratio centrifugal compressors. In the aerospace industry, centrifugal compressors are predominantly used in helicopter engines and small business jet engines, constituting the high pressure-stage in axi-centrifugal compressor configurations. Figure(1.1a) shows a schematic of GE Honda HF120 engine of the Honda business jet, comprising a single-stage fan, 2-stage axial low-pressure

compressor and a single-stage, high-pressure centrifugal compressor. Figure(1.1b) shows a Rolls-Royce-Turbomeca RTM322 helicopter engine comprising a 3-stage axial and single-stage centrifugal compressor.

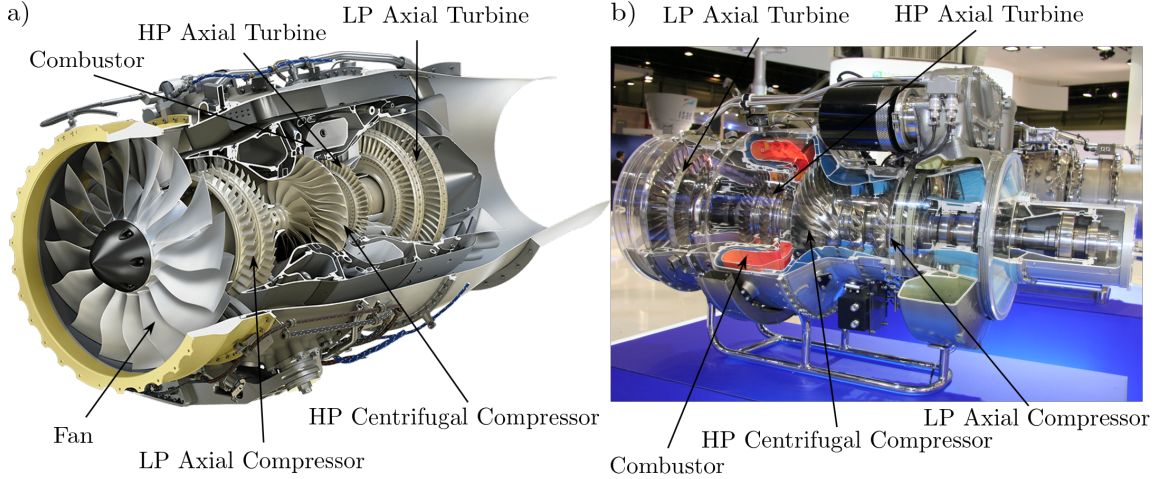


Figure 1.1: a) Schematic of HondaJet HF 120 engine [11] and b) overview of Rolls Royce-Turbomeca RTM322 helicopter engine incorporating axi-centrifugal compressors [12].

1.2 Axial and centrifugal compressor performance

The significant advances in material sciences, manufacturing processes and computational processing power, enabled the design and manufacturing of powerful and highly aerodynamically sophisticated compressor designs. Modern aero-engine compressors are capable of generating pressure ratios which nowadays exceed 50:1 compared to roughly 3:1 of the first aero-engines [13]. For reference, the Rolls-Royce XWB engine generates a pressure ratio of up to 50:1 from an axial-flow, 14-stage compressor, while GEnx is the highest OPR civil aero-engine currently in service, achieving OPR of 58.1:1 at top-of-climb conditions from a 15-stage axial-flow compressor.

Figure(1.2a) shows the pressure characteristics of the NASA High Efficiency Centrifugal Compressor (HECC) for different rotational speeds, ranging from 70% to 105% design speed [14]. It is a centrifugal compressor incorporating an impeller rotor and a vaned diffuser and its intended design purpose is to constitute the high-pressure compressor stage of a rotorcraft aero-engine. Figure(1.2b) shows the characteristic map of a NASA, 10-stage axial-flow compressor [15, 16]. Compared to an axial compressor stage, a centrifugal compressor stage is able to deliver a substantially higher pressure ratio, but is typically less efficient. For reference, modern axial compressors reach efficiencies of over 90% at design conditions, while modern centrifugal compressors do

not exceed 86%-87%. The pressure characteristics of the axial compressor are much steeper, while its mass flow is approximately an order of magnitude higher. The axial compressor shown here is an old design and its efficiency is substantially lower than the one of modern-design, aero-engine compressors.

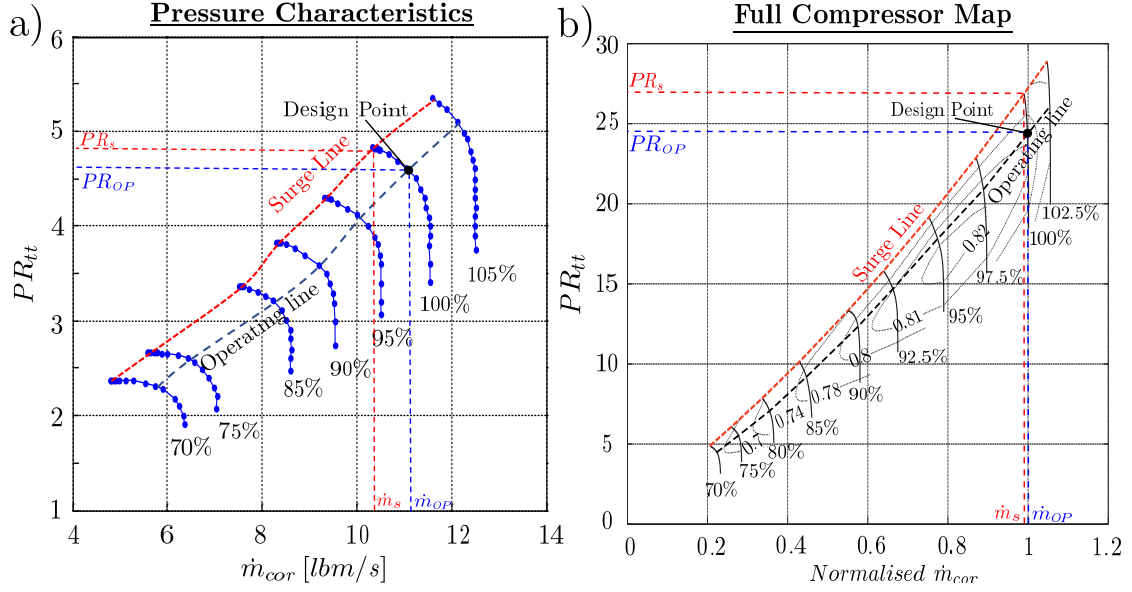


Figure 1.2: a) Pressure ratio characteristics of NASA HECC centrifugal compressor with vaned diffuser [14] and b) pressure ratio and efficiency characteristic of a NASA, 10-stage axial-flow compressor [15, 16].

In compressor passages, the presence of large pressure gradients makes the flow prone to separation. As the mass-flow decreases, the incidence of the incoming flow increases, resulting in higher flow turning and higher energy input into the flow which manifests in the form of higher pressure rise in the lower mass-flow region. The reduction in mass flow is accompanied by a concurrent decrease in the Reynolds number of the aerofoil in the relative FoR, shifting the flow towards the laminar flow regime. This makes the flow more susceptible to flow separation, since the laminar velocity profile is characterised by lower velocities near wall surfaces, thereby making it more susceptible to adverse pressure gradients [17]. The breakdown of the flow causes the blades to stall, substantially increasing losses and preventing further pressurization of the flow. If the mass-flow drops beyond a certain point, then the compressor surges, a phenomenon usually preceded by rotating stall. The implications of those phenomena are catastrophic and might lead to complete loss of power. Figure(1.3) illustrates a F-16 fighter jet suffering compressor stall during the Zeltweg airshow in Austria and a Boeing 777 suffering compressor stall during take-off.

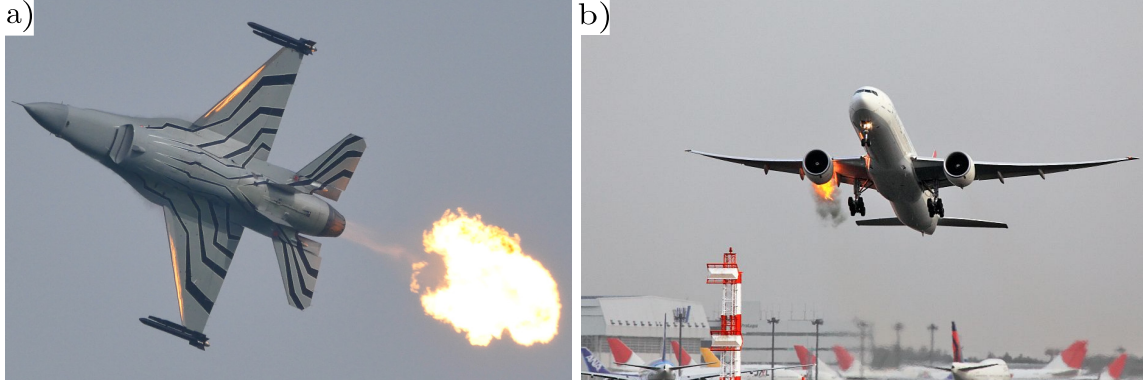


Figure 1.3: a) F-16 suffering compressor stall during an airshow [18] and b) Boeing 777 suffering compressor stall during take-off [19].

On a compressor map, the area of stable operation is delimited by the surge line which indicates the safe range of compressor operation for different rotating speeds and mass flows (see figure(1.2)). To prevent the manifestation of those instabilities throughout the entire flight envelope, the engine is operated within a range far from the surge line, often referred to as *surge margin*. The surge margin ensures that the compressor will be resistive to phenomena which could potentially trigger compressor instabilities. Following NASA's definition [20] the surge margin is expressed as per eqn.(1.1), where the subscripts s and OP, correspond to surge line and instantaneous operating conditions respectively (see figure(1.2)):

$$SM = \left(1 - \frac{PR_{OP}}{PR_s} \frac{\dot{m}_s}{\dot{m}_{OP}}\right) \cdot 100 \quad (1.1)$$

While the surge margin prevents, to a great extent, the manifestation of compressor stall and surge events, exogenous and endogenous situations related to accidents, operability, aerodynamics and mechanical phenomena may lead to the onset of instabilities and potentially compressor stall. There are several phenomena which could trigger instabilities in aero-engine compression systems:

Inlet flow distortions – Inlet flow distortions are generally classified as pressure, temperature and swirl distortions [21]. Pressure non-uniformities at the compressor inlet are commonly encountered at off-design conditions [22], for instance, as a result of flow separation at the lip of the intake, extreme flight manoeuvring or due to crosswinds. Temperature distortions are more commonly encountered in military aircraft due to hot gas ingestion from propelled weapons e.g. missiles or re-ingestion of hot exhaust gases of aircraft with vertical take-off capabilities [21]. The latter is encountered in civil aircraft when the thrust reverser systems are employed.

Swirl distortions typically originate from a compressor experiencing rotating stall, forming a highly non-uniform inlet profile for the downstream positioned compressor, substantially reducing its surge margin. The occurrence of swirl distortions in civil aero-engines during reverse thrust operation was recently investigated by Rajendran and Pachidis [23], using complex CFD RANS and uRANS simulations. Figure(1.4)) shows an overview of the complex flow-field in a high bypass ratio aero-engine during reverse flow thrust operation along with the substantially non-uniform pressure fields at different planes of the engine core inlet.

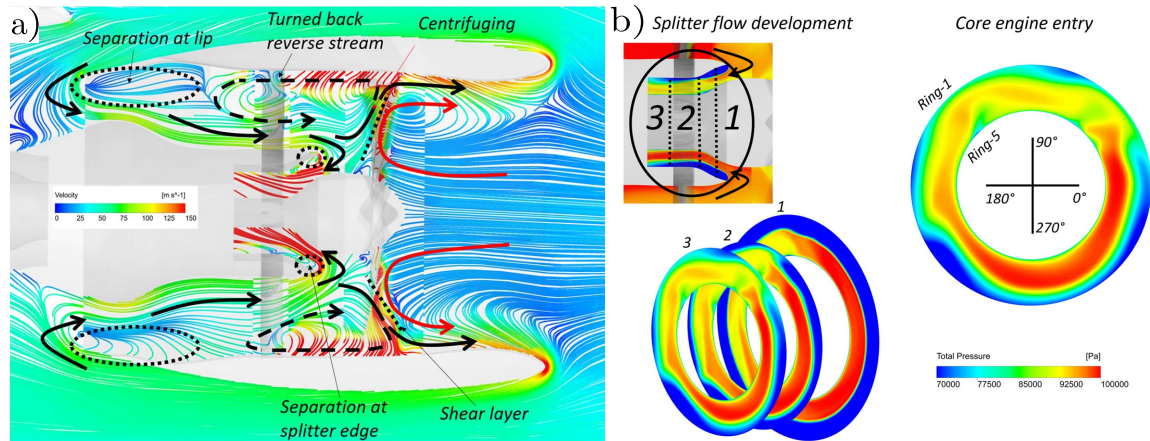


Figure 1.4: a) Overview of the flow-field during reverse thrust operation of an installed, high bypass ratio engine at 110 knots landing speed and b) total pressure contours at different planes at the core inlet [23].

Logic errors in the engine control system – The impact of inlet flow perturbations in the overall engine performance is generally insignificant unless they are intense enough to compromise compressor stability [21]. Even though an inlet perturbation per se, might not compromise engine performance and operability, the propagation of that perturbation from the compressor to downstream engine components might trigger an erroneous action of the engine control system which manages the engine systems by monitoring various pressure and temperature data. Inlet total temperature distortions could for instance affect the estimations of the corrected speed and/or corrected mass flow resulting in an unintended variation in the spool speed. These readings also affect the operation of the variable stator vanes (VSV) and the bleed valves. VSVs effectively change the operating point of the compressor by varying the incidence angle, while bleed valves change the operating point of the compressor by extracting air.

Blade deterioration – Throughout the engine cycle of an engine, the components and primarily compressor and turbine blades undergo extreme thermo-mechanical

loads, causing aerofoil deterioration. There are numerous mechanisms which gradually degrade gas turbine engines e.g. fouling, hot corrosion, oxidation, erosion, damages due to impact of foreign particles and abrasion. Compressor deterioration is primarily driven by tip clearance changes as well as changes in the geometry and surface quality of the aerofoils [24]. Degraded compressor blades exhibit increased losses and consequently reduced efficiencies, since changes in the roughness of an aerofoil modifies the features and the transition point of the boundary layer. This is accompanied by reduced flow turning, causing subsequent stages to operate at off-design conditions, narrowing the stall margins. Compressor deterioration is known to be accompanied by stronger pre-stall disturbances, further compromising their stable operation [25].

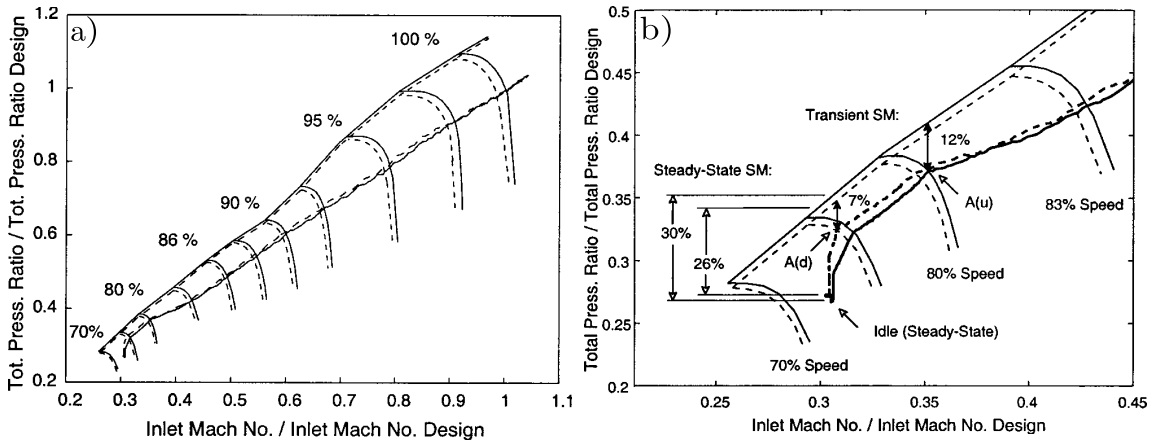


Figure 1.5: a) Compressor maps and transient operating lines of a non-deteriorated (solid) and deteriorated (dashed) compressor and b) overview of surge lines and surge margins between the two compressors at lower speeds [25].

Abrupt change in the operating setting of the engine – Whenever the operating setting of the engine is abruptly varied, the trajectory of the compressor’s operating point temporarily deviates from the operating line [26]. A characteristic example is the so called reslam manoeuvre which comprises an initial sudden deceleration followed by an immediate or delayed acceleration. During rapid transients, there is a hysteresis between the rate of the mass-flow rise in the compressor and the rate of energy-addition in the flow from the compressor blades. The operating point instead of following the trajectory of the operating line, temporarily follows a trajectory on the compressor map which is shifted towards the surge line. This happens because the additional energy input is temporarily imparted to a lower mass-flow air which is converted in pressure rise. If this sudden acceleration or deceleration of the engine drives the operating point beyond the surge line then the compressor might surge. In modern aero-engines such incidences are prevented via an electronic control system

which impedes abrupt changes in the fuel flow.

Shaft failure – Even though shaft failure incidents in aero-engines are generally seldom events, their implications on the jet engine integrity and operability are catastrophic. Shaft failure events may originate to mechanical failure of the shaft itself e.g. due to fatigue and/or other deterioration mechanisms or due to failure of other components. During a shaft failure event, the turbine is decoupled from the compressor and the extracted energy is accelerating the turbine instead of being imparted to the flow via the compressor. The excessive turbine acceleration dramatically increases the stresses due to centrifugal forces; if the turbine speed reaches a critical speed, called terminal speed, then the centrifugal forces are high enough to destroy the disc. The high kinetic energy of the disc fragments is high enough to potentially penetrate the engine cowl (nacelle) causing further damage to the airframe and potentially the passengers.

For certification and safety purposes, engine manufacturers are required to demonstrate the ability of the engine to avoid uncontrolled turbine overspeed [26]. This is achieved through the Full Authority Digital Engine Control (FADEC) control system which receives signals from various instrumentation sensors across the engine. If turbine overspeed is detected, the fuel supply is immediately interrupted and the VSVs close to reduce the mass-flow passing through the compressor and subsequently through the turbine.

During shaft failure, the compressor usually surges. This primarily occurs for two reasons: vibrations in the compression system and mismatch between compressors [27]. The vibrations change the tip clearance causing increased tip clearance to certain blades and reduced to others, reducing the surge margin and making the compressor more prone to instabilities. This is exacerbated if the bearings and the shaft disengage. The second reason for surge inception during shaft failure is mismatch between compressors driven by different shafts. In a multi-stage, multi-pool compressor, if the shaft driving one compressors fails, the mismatch between the compressors might lead the entire compression system to surge.

1.3 Compressor instabilities

The onset of compressor instabilities is prevented by restricting the operation of the engine in a way which ensures that the manifestation, propagation and growth of local flow compressor instabilities will be prevented throughout the entire flight envelope. These instabilities are known as rotating stall and surge and exhibit different

flow patterns [28]. During rotating stall, cells of stalled/reversed flow, rotating at a lower than the rotor rotational speed, form around the annulus resulting in reduced but constant area-averaged mass flow and reduced pressure rise [28]. During rotating stall, the flow-field is highly non-axisymmetric, while the stall cells are rotating around the annulus in the same or the opposite direction to that of the compressor but at a lower rotational speed, typically in the range of 20-70% [29]. Depending on the state of the flow-field as well as the compressor design and operating conditions, a single or multiple stall cells may form around the annulus. Figure(1.6a) shows a schematic representation of a full compressor map which includes forward, reverse and rotating stall characteristics as well as the rotating stall trajectory [30]. A transient evolution of rotating stall cells around the annulus at the TE of the first-stage stator due to change in the angle of VSVs, simulated using 3D, uRANS methods [31] is depicted in figure(1.6).

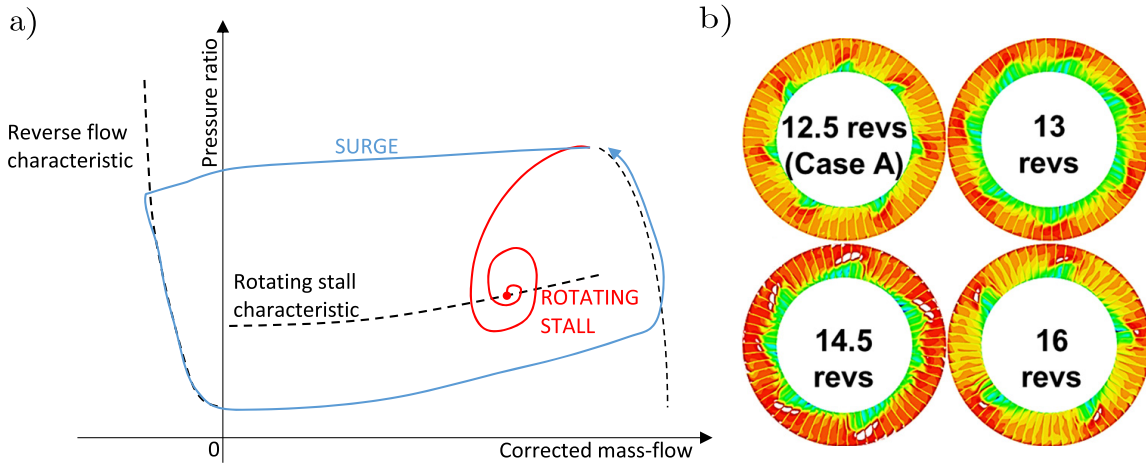


Figure 1.6: a) Indicative compressor map including forward, reverse and rotating stall trajectory and characteristics [30] and b) Transient evolution of different rotating stall patterns during change of VSVs at the trailing edge of a stator vane [31].

Contrary to rotating stall, which is a rather localised instability, affecting certain stages or certain compressors of the compression system e.g. LPC, IPC HPC etc., surge is a global compressor instability phenomenon. It predominantly is an axisymmetric phenomenon characterized by a significant drop in pressure rise and mass flow rate, accompanied by high-amplitude oscillations. Depending on the intensity of the surge event, the onset of extensive flow reversal inside the compressor is possible, a condition often referred to as deep surge [32]. During those events, the intensity and thermo-mechanical loads are high enough to cause catastrophic failure. A catastrophic failure of a small turbocharger due to surge along with the measured pressure traces at the outlet are shown in figure(1.7) [33]. The frequency of the pressure ratio and

mass-flow oscillation during surge are typically in the range 3-10Hz but are dependent upon the entire compression system. These phenomena and the inception mechanisms that trigger their occurrence are up to this date not well understood [28, 34] while the knowledge gap is exacerbated when considering centrifugal or axi-centrifugal configurations [35].

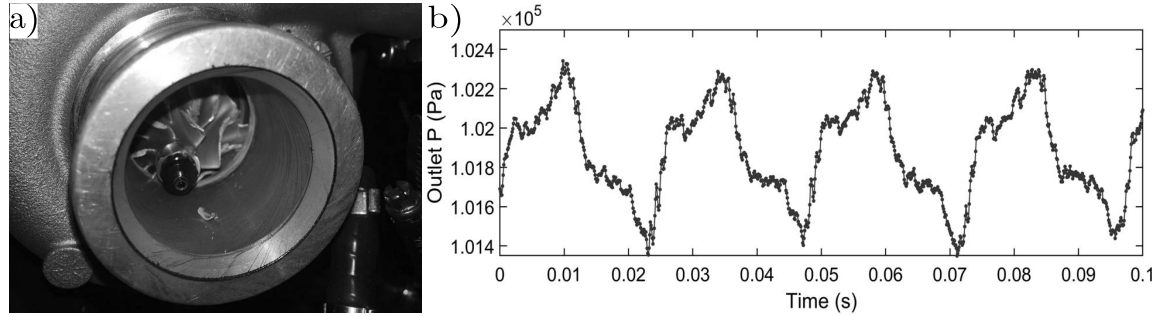


Figure 1.7: a) Catastrophic failure of a turbocharger operating at reverse flow conditions at 135k rpm and b) Surge behaviour of the same compressor at 20k rpm [33].

Sophisticated CFD simulations have demonstrated their potential for accurate reproduction of complex compressor instability phenomena, however the associated computational cost requirements are excessive, making their large-scale, commercial usage challenging. Modelling only half a surge cycle of a full annulus 8-stage modern IPC compressor using URANS CFD could require up to 1 month of computations on a 192-core cluster [36, 37]. Therefore, the development of more efficient methodologies, such as through-flow codes, combining reduced and higher-order methods is highly desirable. Through-flow codes capable of predicting the post-stall aerodynamic response of centrifugal and axi-centrifugal compressors are not found in the literature, while only a handful successful attempts regarding axial compressors is reported. At Cranfield university, a through-flow code for 3D rotating stall and surge modelling for multistage axial-flow compressors has been developed and validated by Righi [27]. An approach adopting and expanding on the primary modelling aspects of that code could provide a novel and computationally efficient framework for the simulation and prediction of axi-centrifugal compressor pre and post-stall behaviour.

1.4 Project aim and objectives

The project aims to develop a body force-based, reduced-order and computationally efficient methodology capable of simulating steady-state and transient aero-engine compressor aerodynamic performance. The realisation of the proposed modelling

approach is performed in the form of a code, encompassing grid generation, solution of the flow-field, body-forces and a combination of reduced-order methods to reproduce the effect of complex flow-phenomena. The overall modelling strategy is expected to be aligned with ACRoSS (Axial Compressor Rotating Stall and Surge) [27], the through-flow code developed within Cranfield University for rotating stall and surge modelling in axial compressors. The project aspires to extend these capabilities to centrifugal compressors by adapting modelling aspects from ACRoSS, to the extent that the limitations and assumptions ensure applicability to centrifugal compressors. The main objectives of the project can be summarised as follows:

- The generation of body force-relevant computational grids in ducts, representative of centrifugal and/or axi-centrifugal compressors. The position of the grid lines should ensure the generation of uniform, non-discontinuous body-force fields in bladed domains. Since the grid size is intrinsically affecting computational efficiency, grid generation should be accomplished in a modular way, allowing the definition of lower and higher mesh resolution areas while preventing abrupt changes in the size of the cells.
- The implementation of adequate numerical schemes to precisely capture essential compressible phenomena such as shock waves. This capability is vital due to the inherently compressible, transonic nature which typically characterises the flow-field in high-speed axial and centrifugal aero-engine compressors. Another essential modelling aspect regarding the solver is the consideration of blade and aerodynamic blockage effects by reducing the effective cell face areas and volumes. This is expected to be an essential modelling aspect at both forward and reverse flow conditions.
- The development of a body force model, based on fundamental principals and universally applicable to all types of compressor blades , i.e. axial rotors and stators, fan blades, impellers, vaned diffuser blades etc. The predictive capability of the model is an essential attribute since it would avoid dependency of the code on CFD or other external inputs.
- The development of a reverse and mixed flow model capable, to the extent possible, of replicating the effect of essential flow features using physics-based, reduced order modelling methods. These flow features entail aerodynamic blockage, flow separation, mixing and losses.

- The evaluation of the validity and potentially the limitations of the individual methods, using numerical test case scenarios, CFD and experimental data. The assessment of the various modelling approaches is essential for the identification of sources of errors and the unveiling of the models' limitations. These limitations could be addressed in the future using different approaches.
- The validation of the code at forward and reverse flow conditions using CFD and if available, experimental data. The code should demonstrate its capability of estimating pre and post-stall aerodynamic performance of axial and centrifugal compressors with reasonable accuracy and minimum external input. Within this project, the academic interest and research efforts are mainly focused on centrifugals compressor modelling, since the axial compressor forward and reverse flow modelling methods have been previously successfully validated by Righi [27, 30, 34, 36].

1.5 Contribution to knowledge

Only two, validated codes with pre and post-stall aerodynamic performance capabilities are reported in the public domain, however their applicability is restricted to purely axial-flow compressors. Regarding centrifugal compressor through-flow modelling, the knowledge gap is dramatically exacerbated; only two, body force-based modelling approaches, applicable to centrifugal compressors exist in the literature. Both of these are subject to significant limitations even at forward flow conditions and are incapable of simulation post-stall compressor response. The present work aspires to develop the first successful, through-flow approach for the modelling of axial and centrifugal compressor pre and post-stall aerodynamics.

To deal with large rotating domains of impeller blade passages, a solver featuring a unique combination of characteristics is developed. A density-based, compressible solver solving the axisymmetric Euler equations in the relative FoR; the Godunov scheme, incorporated in ACROSS, was deemed appropriate due to its robustness and shock-capturing capabilities. The scheme is coupled with 3 Riemann solvers formulated in the relative FoR, while the governing equations with blockage are derived in the relative FoR. A new method, for the evaluation of the additional blockage gradient terms in convoluted ducts is proposed. The terms are precisely computed and the method is not subject to the limitations of other evaluation methods reported in the literature [38]. A new body force model, applicable to all

types of compressor blades (and potentially turbine blades) is developed, comprising a base-turning force, correction force and viscous force component. The new body force model is capable of precisely imposing the prescribed amount of turning in both steady-state and transient conditions, taking into account the full 3D curvature of the blade. The 3D blade curvature is defined via a custom, vector-based analysis. A new distribution coefficient method is proposed, combining streamwise and spanwise data from single-passage RANS simulations to extract the distributions of quantities of interest. The coefficients are subsequently introduced in the code, providing more physically accurate distributions and enabling the reconstruction of complex, viscous phenomena such as boundary layer and mixing-out of tip leakage vortices. The reverse flow modelling approach follows the principle ideas implemented in ACroSS [27] but is further expanded; a new, analytical blockage-mixing model is derived estimating the aerodynamic blockage, mixing losses and flow deviation due to mixing-out of the flow at reverse flow conditions. The developed model is applicable to axial and centrifugal compressor blades, while the aerodynamic blockage is imposed and updated transiently to numerically account for the reduction in the effective area. The consideration of aerodynamic blockage effects, in a transient, physics-based manner is not reported in the literature.

The code is able to predict the pre and post-stall aerodynamic behaviour of axial and centrifugal compressors with reasonable accuracy with the required calibration being limited to a minimum level. Contrary to other through-flow approaches reported in the literature which heavily rely on extensive CFD-derived databases, no CFD input is required, but can optionally be added for more accurate loss and deviation distributions. The ability of the code to simulate deep surge events in centrifugal compressors is also demonstrated and experimentally validated. This is the first successful, validated, through-flow modelling framework for pre and post-stall aerodynamic performance assessment of centrifugal compressors.

1.6 Thesis outline

The first chapter provides the general context of the research topic along with some essential theoretical background on the differences between axial and centrifugal compressors and their operability. Subsequently, phenomena that could trigger compressor instabilities are discussed along with fundamental, introductory concepts regarding rotating stall and surge. The aims and objectives of the project are defined and the projects' contribution to knowledge is discussed.

The second chapter elaborates on the full-lengthscale compressor instabilities, their inception mechanisms and the lower-order modelling approaches available in the literature. The stall inception mechanisms and the phenomena that trigger their occurrence in both axial and centrifugal compressors are discussed. Rotating stall and surge, the mechanisms underpinning their manifestation and the post-stall aerodynamic performance quantification methods in axial and centrifugal compressors are analysed. Reduced-order, modelling approaches, published in the open domain are presented, comprising 1D, 2D streamline curvature methods and body force-based methods, as well as 3D, though-flow approaches. Their maturity, capabilities and deficiencies are thoroughly assessed. Modelling concepts for blockage effects, body forces, essential aspects for forward and stalled flow such as losses and flow deviation are analysed to evaluate their limitations and applicability within the context of this work. Based on the findings of the literature review process, the research gap is identified.

The third chapter describes the various modelling layers embedded in the code, accompanied by the theoretical background of the models and the fundamental principles their derivation is based upon. Detailed derivations of the models are provided in the appendices. The main topics include the generation of the computational domain, the solver and the governing equations, the blade curvature definition method, the body force model, the blockage-mixing model at reverse flow conditions, the empirical loss and deviation models, the forward, mixed and reverse flow modelling approach and the plenum-throttle model.

The fourth chapter essentially incorporates the validation process of the individual modelling layers of the developed methodology. The first stage of the validation process includes the validation of the solver, the validation of the body force model and the validation of the curvature extraction method. The validity of the solver is performed by validating and comparing different Riemann solvers against 1D, Sod shock tube-type of problems and comparing the flow-field solution of the code against the CFD one in empty duct simulations. Test cases are performed to assess the validity of the modified equations with blockage and their implementation. The capabilities and limitations of the body force components is individually evaluated using numerical test scenarios and different blade geometries, representative of axial and centrifugal compressor rotors and stators. The turning force model is assessed by comparing the misalignment of the flow with respect to the imposed flow path. The validity and limitations of the viscous force model is evaluated by comparing the prescribed and imposed losses in an axial rotor and an impeller passage. The

distribution coefficient concept is validated by comparing the prescribed pressure losses and flow angles against the averaged measured ones, when different coefficients are imposed. A preliminary verification of the reverse flow modelling method and the blockage-mixing model is carried out by comparing losses against CFD data and aerodynamic blockage predictions against other models reported in the literature.

The fifth chapter comprises the validation of the entire framework and is discussed in two subsections; forward flow and reverse flow. The forward flow validation is performed using a combination of experimental and CFD data of one axial rotor, one axial-flow fan stage and 3 centrifugal compressors with vaneless and vaned diffusers. Global, steady-state, aerodynamic performance predictions along with flow-field solution comparisons against CFD results provide a quantitative and qualitative validation of the code. The steady-state verification process at reverse flow conditions is performed entirely using CFD, due to lack of experimental data. The ability of the code to provide realistic transient (surge) behaviour is assessed in 2 centrifugal compressors configurations by varying the plenum volume and is validated against experimental results.

The sixth chapter provides an extended summary of the thesis, summarises the most essential conclusions of the present work and is concluded with the a detailed overview of the contributions to knowledge and the author's recommendations for future work.

Chapter 2

Literature Review

This chapter provides a theoretical background on the full-lengthscale instabilities in both axial and centrifugal compressors and their inception mechanisms and further expands on different, reduced-order modelling approaches, elaborating on their capabilities, limitations and individual modelling strategies. The mechanisms that trigger the occurrence of spike and modal stall precursors in axial and centrifugal compressors are discussed along with their salient attributes and differences. The implications of full-lengthscale instabilities, known as rotating stall and surge, on the response of axial and centrifugal compressor systems are then analysed. The advantages and limitations of modelling methods of different levels of fidelity are explored, including 1D, 2D and 3D through-flow methods for axial and centrifugal compressor pre and post-stall modelling. Body force methods are split into 4 main categories and their capabilities and limitations are discussed. The chapter is concluded with the analysis of certain reverse flow modelling approaches reported in the literature. While a limited number of validated, body force-based approaches for axial compressor rotating stall and surge modelling exists in the literature, none are reported for centrifugal compressors.

Contents

2.1	Stall inception	18
2.2	Compressor stall	24
2.3	Compressor aerodynamic performance modelling methods	36
2.4	Lower-order modelling methods	48
2.5	Research gap	65
2.6	Chapter summary	67

2.1 Stall inception

Rotating-stall and surge constitute the mature, full-scale instabilities that a compression system might experience once a compressor stalls. The onset of those phenomena is preceded by perturbations, which depending on the operating conditions of the compressor and the flow-field, will either grow to full-scale instabilities or might decay over time without the compressor eventually going into stall. These perturbations are known as *modal waves* and *spikes* [39] and their manifestation is a function of both the compression system and the flow-field conditions.

Modal waves are long-wavelength perturbations, characterised by smaller amplitude than spikes and are typically occupying a large circumferential extent of the annulus. The lengthscale of the waves is often expressed with respect to the blade pitch; a short-lengthscale perturbation typically corresponds to a few blade pitches (e.g. 1,2 blade passages) while a long-lengthscale perturbation indicates a phenomenon occupying a large fraction of the circumferential extent, thereby covering multiple blade passages and potentially the entire annulus [40]. Modal oscillations are further categorised according to their wavelength, with a first mode indicating a perturbation occupying circumferentially the entire annulus, second mode half of the annulus etc. In lower speed compressors, modal waves typically rotate at a speed in the range [20%-50%], [39, 41]. Modal waves generally are not localised phenomena (streamwise); they usually propagate along the entire compression system in the meridional direction, although Camp and Day report that stage mismatching in axial compressors can axially confine the propagation of modal waves in axial compressors [39]. Depending on the averaged background flow-conditions, modal waves could be damped without destabilizing the compressor or they could be amplified if the dampening of the background flow becomes negative. The theoretical interconnection between the dampening or amplification of long-wave length disturbances preceding stall inception was first predicted by the Moore-Greitzer model [42] and was later experimentally confirmed by McDougall et al. [43].

The second type of stall inception precursors are known as *spikes*. These are short-lengthscale disturbances and unlike modal waves, which manifest and progressively grow before stall, they arise suddenly, causing the compressor to experience rotating stall or surge shortly after their appearance [39]. This stall inception mechanism is usually related to three-dimensional breakdown of the flow, usually near the rotor

shroud of a single passage. They initially manifest in one blade passages due to flow separation which causes increased aerodynamic blockage. The blockage causes the incoming flow to divert, increasing and reducing the incidence angle of adjacent blades thus propagating circumferentially. A schematic overview of the spike-stall development based on Emmons [29] is shown in figure(2.1). For many years, it was hypothesized that spike stall inception is purely related to tip clearance, thereby constituting a stall inception mechanism which could manifest exclusively in unshrouded compressors. It was proven later, that shrouded axial-flow compressors could also experience this kind of perturbations [44], while more recently, Spakovszky and Roduner demonstrated the presence spike instabilities in centrifugal compressors [35]. Due to the small circumferential length of spikes, they propagate rapidly around the annulus, usually at a speed between 60%-80% of the rotor [39]. Short-lengthscale perturbations, hundreds of rotor revolutions before the stall of an axial, four-stage GE compressor were identified by Park [45]. These were emerging and decaying during the pre-stall operation of the compressor and the compressor stalled once the amplitude of the spike was sufficiently high to destabilise it.

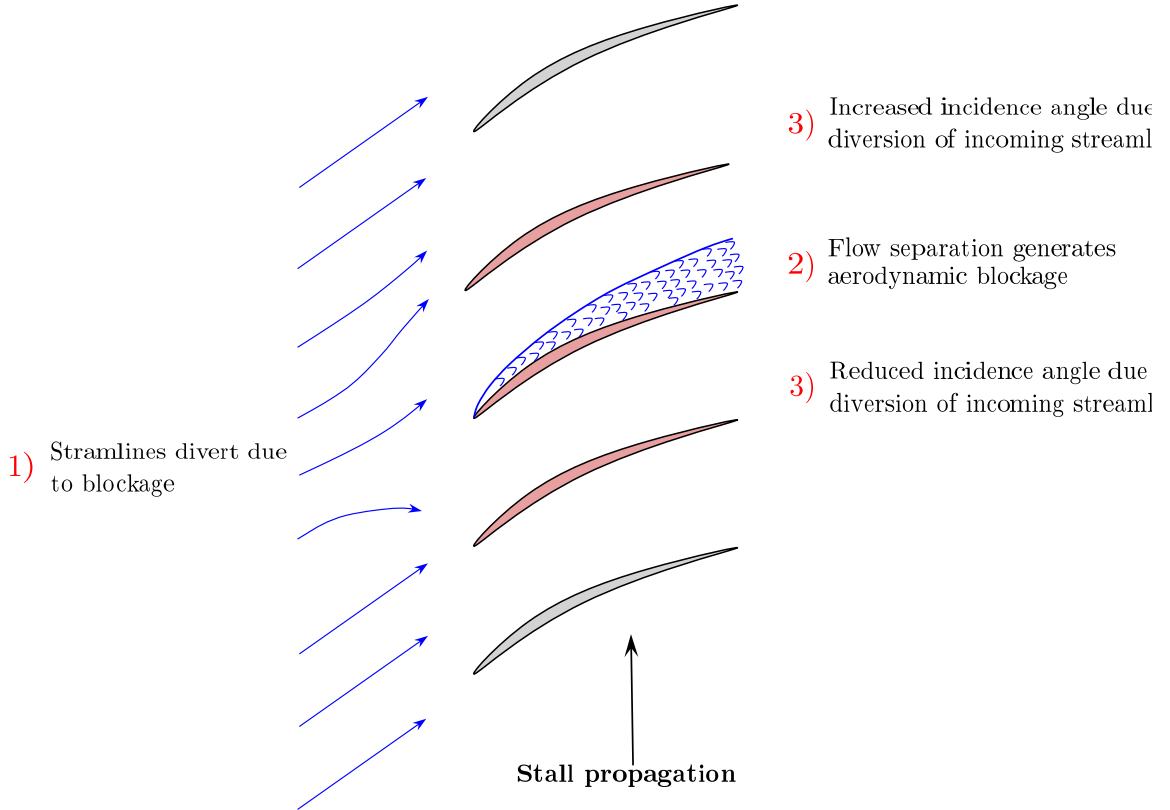


Figure 2.1: Illustration of the circumferential propagation of a spike instability due to flow separation and change in the incidence angles of adjacent blades, based on Emmons [29].

Modal waves and spikes are precursors of stall inception in both axial and centrifugal compressors. Camp and Day [39] investigated the stall inception mechanisms of the C106 compressor in the Whittle Laboratory (Cambridge), a low-speed, 4-stage axial-flow compressor. Both modal waves and spike precursors were experimentally identified while the manifestation of the type of instability was concluded to be primarily dependent upon a *critical rotor incidence angle*. Depending on the inlet guide vane (IGV) position, it was demonstrated that the compressor could stall either due to modal waves or due to spikes. Spikes were arising when the critical rotor incidence was exceeded; this was causing a breakdown of the flow near the shroud in a few blade rotors, which was quickly propagating circumferentially causing the compressor to stall. Modal waves were recorded near the peak of the total-to-static characteristic; when the modal oscillations caused the incidence angle of a particular rotor to exceed the critical one, spikes would manifest and weak modal waves would coexist with spikes. In that case, the inception of the stall initiated due to the short-wavelength disturbances (spikes) and not the modal waves. When modal oscillations didn't induce incidence angles at the rotors higher than the critical one, then the compressor was reported to stall due to the gradual amplification of the modal waves, causing separation to a larger number of blade passages which subsequently evolved into a rotating stall cell.

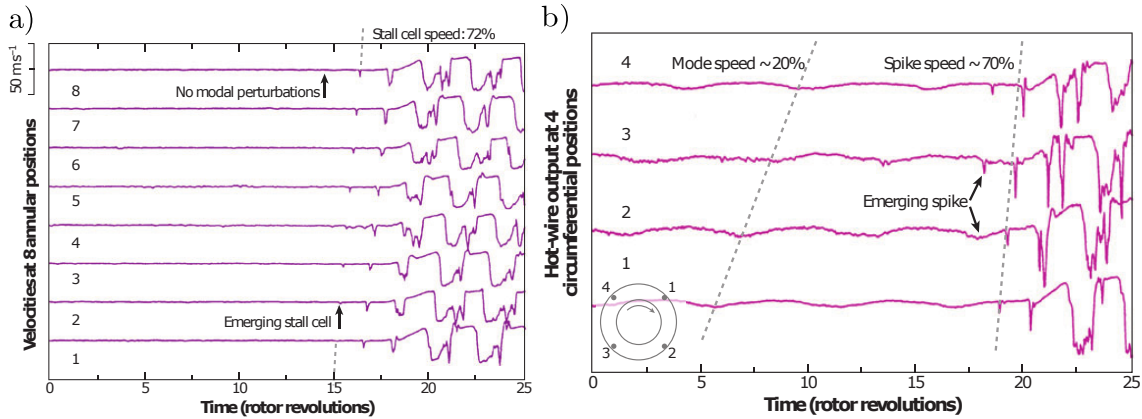


Figure 2.2: a) Example of a spike disturbance in a low speed, four-stage axial-flow compressor, (original data by [46] and adapted from [47]) and b) coexistence of weak modal waves and spikes in the same compressor (original data from [39] and adapted by [47]). The compressor in the second case stalls due to spikes.

Figure(2.2a) shows the emergence of spike disturbances preceding the onset of a rotating stall event, rotating at 72% of the shaft speed in the C106 compressor [46]. Figure(2.2b) shows the coexistence of a weak, low amplitude modal wave rotating at 20% of the rotor speed and the emergence of a spike instability shortly after, rotating

at 70% of the rotor speed in the same compressor. Even though compressor stall is preceded by the emergence of both instabilities, the compressor stalls due to spikes, which in turn are triggered by the modal waves as described earlier.

Based on observations of their experimental campaign, Camp and Day [39] proposed a criterion, linking the peak of the pressure rise characteristic with the critical incidence angle. According to their criterion, if the rotor incidence angle exceeds the critical one before the compressor reaches the peak of the overall pressure characteristic, then the compressor will stall due to spikes and the spike will emerge in the most highly loaded rotor. If the peak of the pressure characteristic is reached without the incidence angles exceeding the critical one, then the compressor will stall due to modal waves. This is schematically illustrated in figure(2.3).

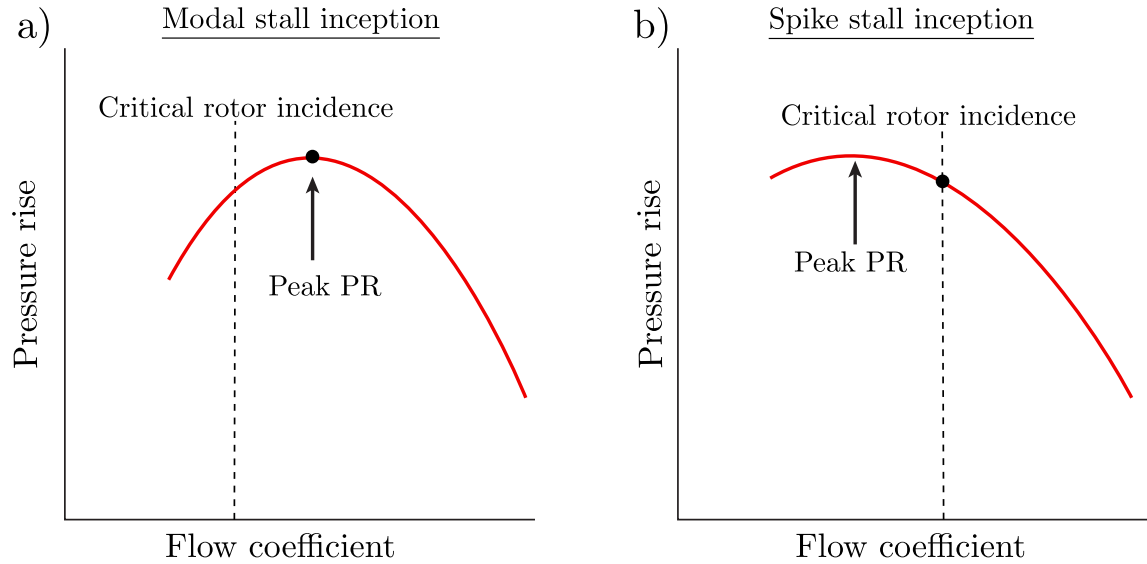


Figure 2.3: Stall inception criterion of Camp and Day for axial-flow compressors based on the critical rotor incidence angle and the peak efficiency point of the pressure characteristic [39, 47].

Spakovszky and Roduner [35] experimentally investigated the stall inception mechanisms in a turbocharger centrifugal compressor with vaned diffuser intended for automotive applications. A bleed valve was located just downstream the impeller outlet and data were obtained with and without bleed flow. Without bleed flow, the compressor at both 100% and 105% corrected speed surged due to the appearance of short-wavelength disturbances (spikes) in the vaneless space (figure(2.4a)). The bleed flow reduced the endwall blockage in the vaneless space caused by the low momentum boundary layer, increasing the pressure rise in the vaneless space but destabilising the highly-loaded vaned diffuser. With bleed flow, the compressor stalled due to the appearance of modal waves which manifested approximately 80

revolutions prior to surge. The vaneless and the semi-vaneless space were identified as the critical/destabilising components of the compressor.

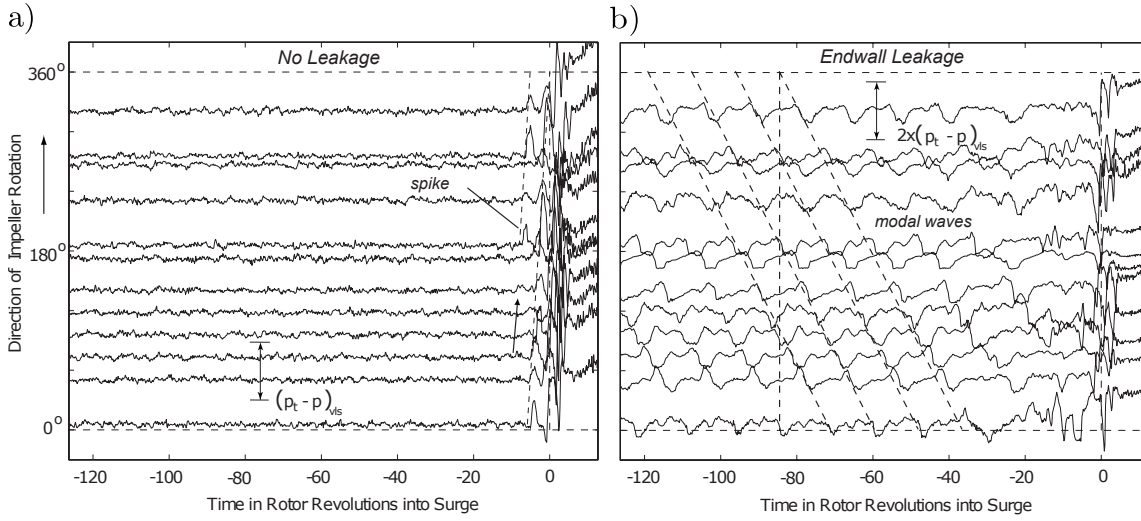


Figure 2.4: Pressure measurements at 105% corrected speed in the vaneless space of a centrifugal compressor with vaned diffuser a) without endwall leakage stall occurs due to spikes and b) with endwall leakage stall occurs due to modal waves [35].

In a similar study [48], a CFD model of the same compressor was used in order to conduct numerically-forced response experiments using unsteady, 3D RANS simulations aiming to investigate the type of the stall precursors leading to compressor instability. The simulations were performed using an isolated, full annulus diffuser, while pitchwise averaged boundary conditions were specified at the inlet, obtained from single passage, RANS simulations. To destabilise the diffuser, a disturbance by means of an increase in total pressure was applied between 80% span and the shroud, for a time interval equal to 1/4 revolution. At design speed, this resulted in a large incidence angle at the diffuser LE close to the shroud, due to the high spanwise non-uniformity of the flow. This led to the formation of a separation bubble and reverse radial flow. Vorticity was convected back to the vaneless space (figure(2.5b)) enhancing flow blockage which in turn led to the appearance of short-wavelength disturbances (spikes), matching the aforementioned experimental results. At 78% speed, where the flow was spanwise more uniform, no instability could be triggered. Simulations using body force models [49] indicate that, at these conditions, the compressor would stall due to modal waves. Modal waves cannot be captured from the isolated diffuser model, since modal waves are inherently related to the interaction between the impeller and the diffuser. It was concluded that spikes are associated with high spanwise non-uniformity at the impeller exit while the compressor stalls via modal waves when the flow is spanwise more uniform. The importance of impeller

and vaned diffuser interaction was demonstrated by Spakovszky [50], who, using a lower-order, analytical model along with experimental data, demonstrated that the interaction of pressure waves and unsteady vortex shedding is driven by the interaction between the impeller and the vaned diffuser. This gave rise to backwards-travelling rotating stall waves destabilizing the vaneless space between the impeller and the vaned diffuser.

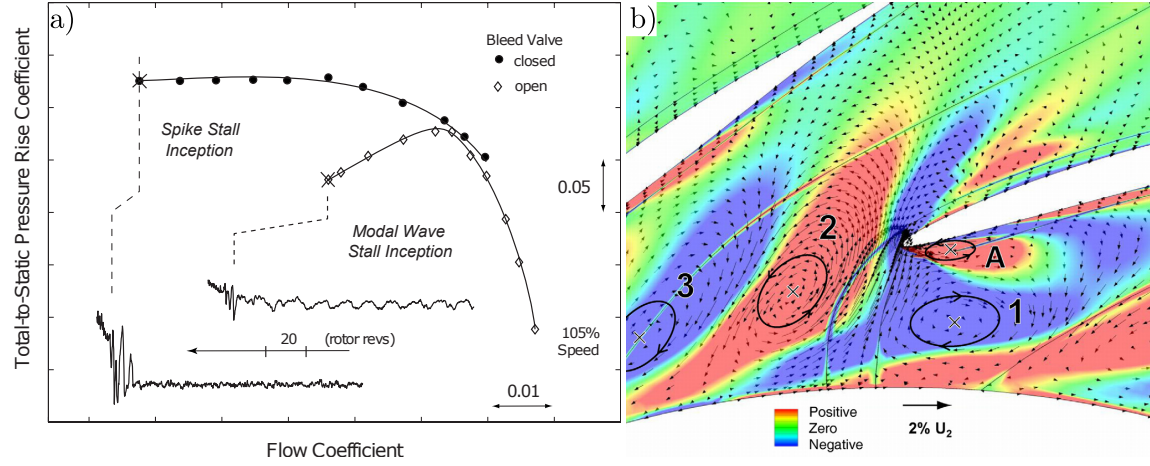


Figure 2.5: a) Overview of compressor characteristics for the 2 different stall inception mechanisms: spikes and modal waves [35] and b) counter-rotating vortices convecting in vaneless space generating blockage [48].

Spakovszky and Roduner [35] in their study state, that the criterion for modal and spike-stall inception of Camp and Day seems to also underpin the behaviour of centrifugal compressors. Based on the results of their study, they proposed a preliminary criterion for stall inception in centrifugal compressors with vaned diffusers, according to which, the compressor is expected to stall due to modal waves if the slope of the semi-vaneless space characteristic becomes zero before the peak of the overall diffuser static pressure characteristic. Figure (2.5a) illustrates the different paths to instability in a centrifugal compressor, stalling due to modal waves and spikes. Short-wavelength instabilities initiate stall almost immediately and this usually occurs at the point of the characteristic where the pressure ratio curve reaches a plateau. On the contrary, when the second type of instability occurs, the slope of the characteristic reaches its peak pressure ratio with the characteristic changing from a negative to a positive gradient before the compressor stalls. In centrifugal compressors with vaned diffusers, the destabilizing component is usually the vaned diffuser [48].

2.2 Compressor stall

The onset of compressor stall initiates whenever the amplitude of the long-wavelength or short-wavelength disturbances is large enough to evolve into a full-lengthscale instability, eventually causing the compressor to stall. Depending on the operating conditions of the compressor and the flow-field, two types of compressor stall can occur: *rotating stall* and *surge*. Typically, rotating stall is preceding a surge event, however a surge event can occur without the prior emergence of a rotating stall event. Rotating stall is not an irreversible instability; it is possible for a rotating stall pattern, consisting of one or more stall cells to dissipate, allowing the compressor to recover.

2.2.1 Rotating stall

Rotating stall is a phenomenon whereby one or more circumferential sectors of the compressor annulus exhibit stalled flow characteristics, rotating around the annulus at a fraction of the rotor speed. Rotating stall is a dynamic, transient phenomenon therefore the number and the size of the stall cells change. At its fully developed form, often referred to as mature or fully developed rotating stall, the operating conditions of the compressor e.g. mass flow and pressure ratio are constant and distinct sectors of stall cells are present [27]. During rotating stall, the overall mass flow of the compressor is significantly lower, while the efficiency might drop even below 20% during those conditions as reported by Iura et al. [51]. The frequency of a rotating stall event is usually in the range 50-100 Hz which is approximately an order of magnitude higher than the frequency of a surge event [52]. As the operating point moves along the rotating stall characteristic, different rotating stall patterns form around the annulus which usually merge into a larger, fully developed stall cell as the mass flow keeps decreasing. Iura et al. [51] observed different rotating stall patterns in a low-speed, 3-stage, axial-flow compressor including part-stall and full-stall patterns. In the higher flow coefficient range, a part-stall was observed, i.e. the stall cells were not radially occupying the entire blade passage (from hub to tip), while in the lower, full-span stall was reported. A torque and exit duct wall pressure of a low-speed, three-stage, axial-flow compressor is shown in figure(2.6) along with overview of the rotating stall cells at different flow coefficients. At a flow coefficient of approximately 0.4, a small stall cell emerged, rotating at 38% of the rotor velocity. Further reduction in mass-flow caused the formation of different patterns, which were stable but very sensitive to throttle changes. The formation of a three-stall cell was

reported when the flow coefficient was reduced to 0.36, while further reduction of the flow coefficient to 0.323 caused the multi-stall cell patterns to merge into a single, large cell rotating at approximately 26% of the rotor speed.

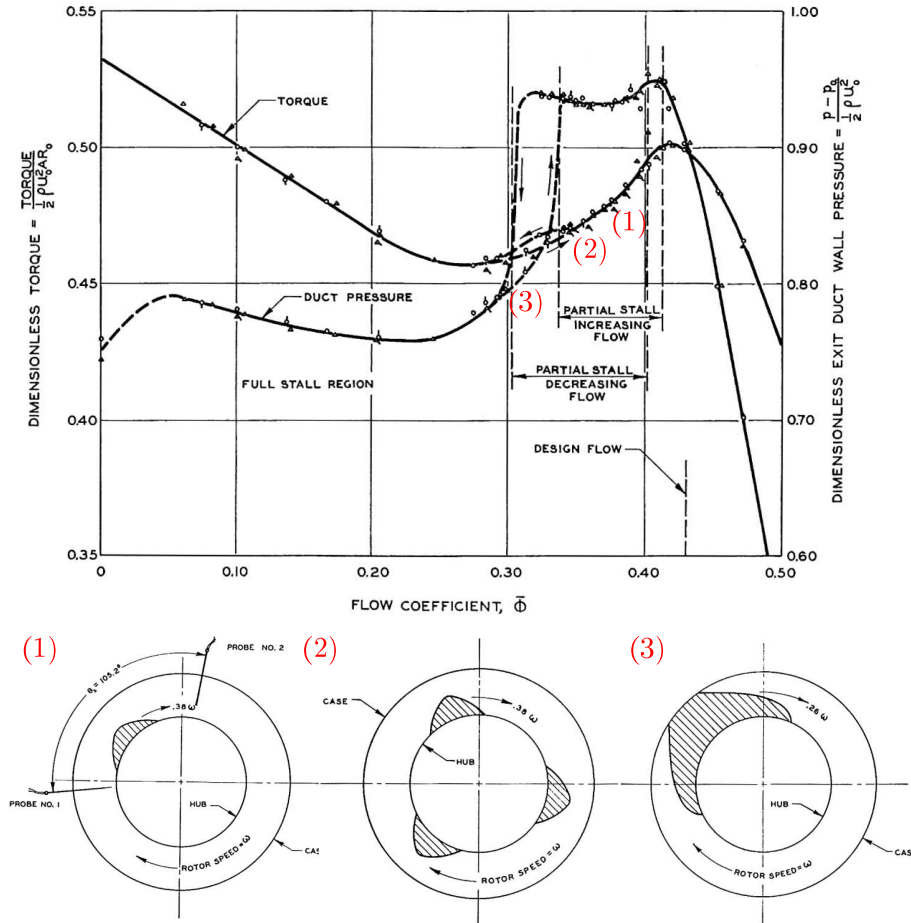


Figure 2.6: Torque and exit duct wall pressure characteristics of a low-speed, three-stage, axial-flow compressor and overview of the rotating stall cells patterns at three flow coefficients (adapted from [51])

Day and Cumpsty [53] were the first who provided a more accurate insight into the flow inside a stall cell using fast response instrumentation. They concluded that the axial component of the flow (in axial compressors) is minuscule, while the flow in the unstalled regions reaches velocities higher than the corresponding design values. The flow inside a stall cell is rather active and three-dimensional due to centrifugal forces which increase the pressure near the shroud. This is in contrast to the misconceptions of the earlier days, when the stalled flow was considered as a "dead wake". It was also observed that the tangential velocity of the flow inside a stall cell is remarkably high, while upstream the rotor, it can even exceed the rotational speed of the wheel. The flow is convected between the boundaries of the stall cells by the rotors, as demonstrated in figure(2.7).

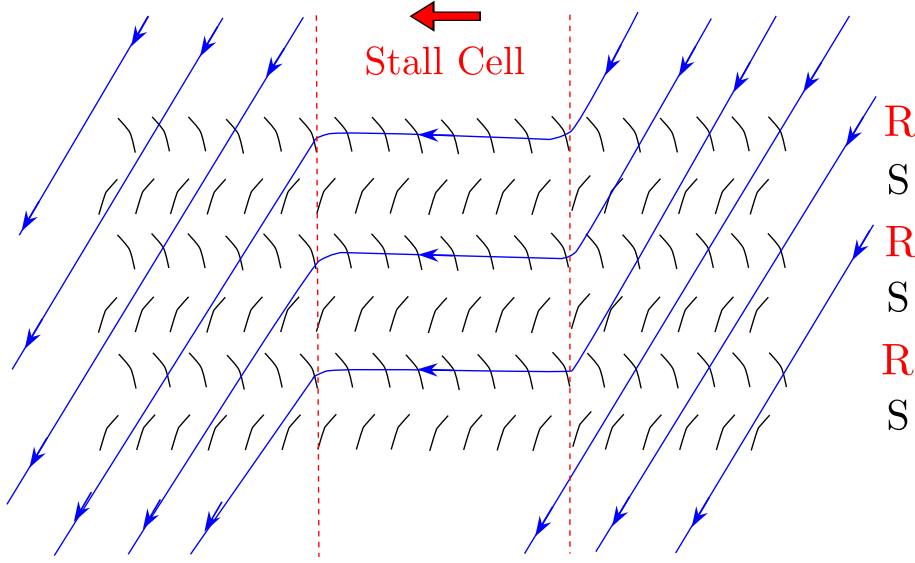


Figure 2.7: Flow inside and outside stall cell. The flow inside the stall cell is convected circumferentially through the rotors (adapted from [28]).

Rotating stall is not a phenomenon limited to axial-flow compressors. Rotating stall in impeller, vaneless space as well as vaneless and vaned diffuser is reported by [54]. Frigne et al. [55] investigated experimentally an impeller with a vaneless diffuser and reported different rotating stall patterns both in the impeller and the vaneless diffuser (figure(2.8 a,b)). The diffuser inlet flow angle, denoted by α_2 , was concluded to be the key parameter for the stability of the diffuser and dictated the characteristics of the rotating stall pattern. At $\alpha_2 = 78^\circ$ (higher α implies lower mass flow), 4-5 stall cells rotating at 14% the impeller speed were recorded. The velocity fluctuations were approximately 6.5% of the mean velocity and the phenomenon was therefore characterised as mild impeller rotating stall (MIRS). At $\alpha_2 = 81.5^\circ$, 1-3 rotating stall cells emerged, with the velocity fluctuations being in the order of 30% of the mean velocity. Due to the large fluctuations and the sudden emergence of the cells, the phenomenon was characterised as abrupt impeller rotating stall (AIRS). Using a slightly different configuration, at $\alpha_2 = 76^\circ$, rotating stall was observed at both the impeller and diffuser inlet with the velocity oscillation amplitude progressively increasing from 5% to 10% and was therefore called progressive impeller rotating stall (PIRS). By applying 40 degrees pre-rotation and unloading the impeller, rotating stall in the vaneless diffuser was recorded when the inlet diffuser angle was set to $\alpha_2 = 77^\circ$. The rotating stall event exhibited 3 cells rotating between 17% - 21% of the impeller velocity. Figure(2.8a) shows the pressure rise coefficient of the different compressor components at different inlet diffuser angles, including the number of stall cells (denoted by λ), while figure(2.8b) shows the different rotating stall patterns at

different rotational speeds and diffuser angles.

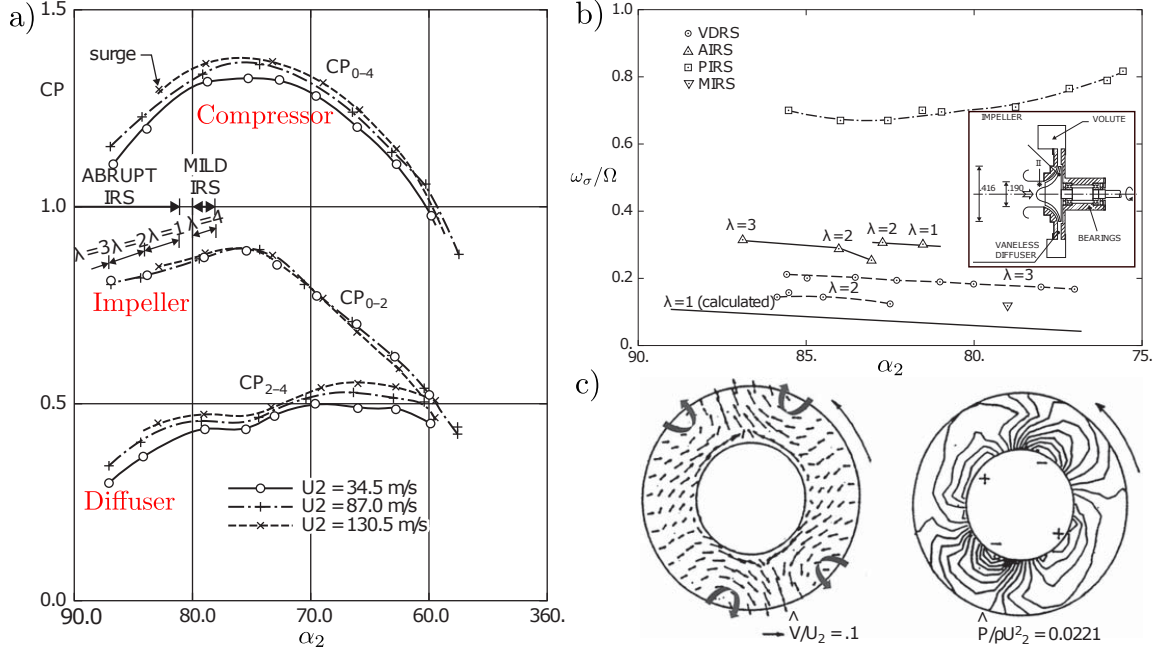


Figure 2.8: a) Pressure rise coefficient in the entire compressor, impeller and vaneless diffuser, at different diffuser inlet angles before and after mild or abrupt rotating stall [55], b) different types of impeller rotating stall and number of cells at different rotational speeds [55] and c) vaneless diffuser rotating stall experimental measurements ([54]).

Vaneless diffuser rotating stall was first mentioned by Jansen [56] who demonstrated that local reversal of the radial velocity component in the vaneless diffuser could precipitate rotating stall events in a vaneless diffuser. Figure(2.8c) shows the distribution of the measured velocity and pressure during a rotating stall event in a vaneless diffuser [55]. Areas of low velocity (stall cells) and high velocity can be seen along with the presence of two, counter-rotating vortices. The radial velocity in the unstalled region is substantially higher than in normal operating conditions. A similar pattern in a vaneless diffuser was also reported by Wachter and Rieder [54]. Rotating stall was reported by Oakes et al. [57] who successfully used air injection for active stall control in a low-speed, mixed-flow impeller with vaned diffuser to recover the compressor from a rotating stall event.

Munari et al. [32] experimentally investigated the pre and post-stall aerodynamic performance of an Allison, axi-centrifugal compressor used in the turboshaft 250-C18 engine. They reported a rotating stall cell rotating at 13% of the rotor speed and its emergence was associated with the axial stages which were the ones destabilising the compressor. The size of the stall cell was increasing as the compressor was approaching surge, while alternation between rotating stall and surge events was

observed after the occurrence of surge.

2.2.2 Surge

Surge is an axisymmetric phenomenon, affecting the entire compression system. During a rotating stall event, the flow is circumferentially non-uniform due to the presence of stall cells of low velocity rotating around the annulus, however the average mass flow and pressure ratio are constant. During a surge event, the mass flow and the pressure ratio constantly oscillate at frequencies which typically are an order of magnitude lower than the rotating stall, e.g. in the range 3-10 Hz [52]. Depending on the severity of the surge event, the mass flow might undergo oscillations without the average mass flow becoming negative; in that case the surge event is characterised as classic surge. If during the surge event, large-scale flow reversal occurs i.e. negative mass flow, then the event is classified as deep surge [27, 58]. The type of instability developed in a compressor as well as the intensity of the surge phenomenon (classic or deep) depends largely on the entire volume of the compression system; typically a compressor attached to a large volume tends to surge, while the compressor tends to stall if it is attached to a small volume, e.g. short length pipe [28].

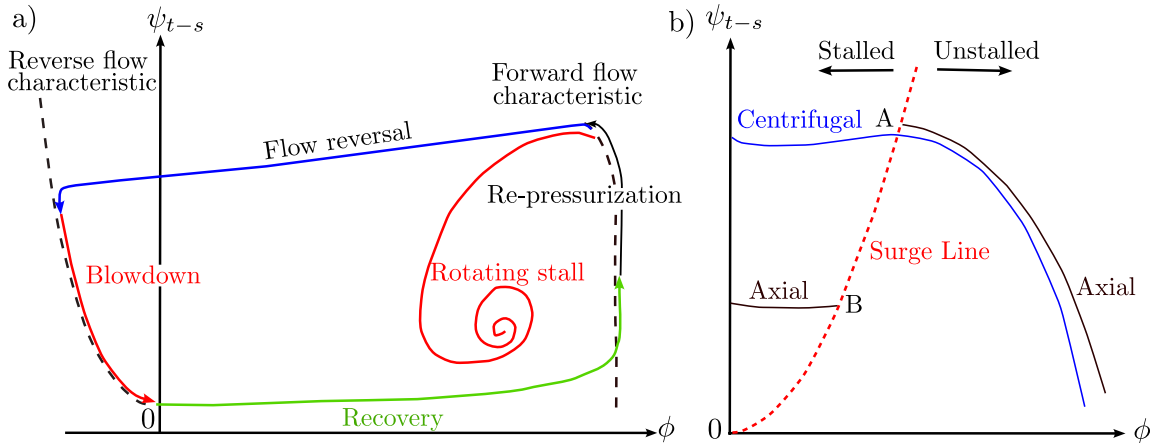


Figure 2.9: a) Post-stall deep surge and rotating stall trajectories and the four stages of a deep-surge cycle (adapted from [58]) and b) comparison of post-stall trajectory in a high-speed axial and centrifugal compressor (adapted from [28]).

Surge events are periodic phenomena typically comprising multiple cycles. During a surge cycle, the compressor will undergo four distinct phases (figure(2.9a)): *reversal*, *blowdown*, *recovery* and *re-pressurization* [58]. Highly loaded, high-pressure ratio axial compressors experience large pressure drops during rotating stall, which typically are in the order of 50%. On the contrary, low-pressure ratio axial machines and centrifugal compressors normally exhibit a significantly less discontinuous behaviour beyond the

surge line. This was first noted and experimentally identified by Emmons [29]. The difference between the post-stall behaviour of axial and centrifugal compressors is schematically illustrated in figure(2.9b) showing two notional extended characteristics of an axial and centrifugal compressor. The large discontinuity in the axial compressor pressure characteristic corresponds to the rotating stall region. The rotating stall region of the centrifugal compressor characteristic is continuous and does exhibit only a mild pressure loss. During a deep surge cycle, the compressor undergoes four distinct phases:

- **Flow reversal/rotating stall** – Once the operating point of the compressor moves beyond the surge line and the compressor stalls, depending on the conditions mentioned earlier, the compressor will either follow the rotating stall trajectory, surge or both. Flow reversal is a rapid, transient phenomenon whereby the direction of the flow in the compressor switches from positive to negative in a few rotor revolutions. Steady-state operation is experimentally and numerically feasible along the rotating stall characteristic but not in the flow reversal region.
- **Blowdown** – During the blowdown phase of a surge cycle, reverse flow is dominating the compressor, moving from the plenum towards the physical inlet of the compressor. This results in a progressive reduction in the plenum's static pressure. The pressure delivery of the compressor keeps diminishing continuously while the mass-flow progressively reduces, approaching zero. Steady-state operation in the blowdown region is experimentally and numerically feasible, with the steady-state points essentially forming the *reverse flow characteristic*.
- **Recovery** – Once the pressure in the plenum reaches a certain value, the substantial reduction in the pressure gradient across the duct enables the compressor to accelerate the flow in the forward direction. During the recovery phase, the flow inside the blade passages gradually unstalls, resulting in a concurrent increase in both the mass flow and the pressure ratio. Similar to the flow reversal phase, this is a highly transient region and no steady-state operation is possible.
- **Re-pressurisation** – The compressor subsequently completely unstalls and starts moving towards the speed characteristic it followed prior to the surge

event. Due to the highly de-pressurized conditions in the plenum, the compressor initially chokes. The pressurization process of the plenum then initiates, continuously shifting the operating point of the compressor upwards, towards the surge line. If the throttle valve remains closed and the pressurisation of the plenum continues, the compressor will eventually surge again and the cycle will repeat.

At MIT, Gamache and Greitzer [59] experimentally investigated mature rotating stall and steady-state reverse flow conditions using a 3-stage, low-speed compressor. The compressor flow-path was of constant-area design, the hub-to-tip radius was 0.88 and the compressor was driven by an electric motor. Downstream the compressor, a plenum chamber was attached along with an adjustable conical nozzle via which, the mass-flow was adjusted. Mass-flow measurements were obtained via an orifice plate located downstream the plenum which was preceded by a series of porous screens and honeycomb plates to de-swirl the incoming flow before it reached the orifice plate. Measurements were obtained in planes located in the intermediate duct regions, in-between the blades as well as upstream and downstream the last stator and IGV respectively (downstream with respect to reverse flow conditions). Measurements were taken at eleven circumferential positions and 5 radial planes. The experiments were performed for 2 configurations by re-staggering the rotor and the stagger angles: a high-reaction and a low-reaction one. Both configurations exhibited rotating stall events which preceded compressor surge. Steady-state, rotating stall operating points were obtained for a range of flow coefficients including near-zero net mass-flow. The rotating stall cell in the lowest mass flow condition was occupying approximately 90% of the compressor's circumferential extent, while the transition from rotating stall to surge was accompanied by a pressure rise and torque discontinuity in the order of 60%. The total-to-static pressure rise and torque coefficient characteristics are shown in figure(2.10).

The high-response measurements helped elucidate the flow-field structure inside blade passages during reverse flow-conditions. Gamache and Greitzer report the formation of a *jet-wake structure*, due to the highly separated flow in the suction side of the blade caused by the excessive incidence angles of the incoming flow relative to the blade's TE angle which was reported to exceed in several occasions 90° . The significance of the mixing process in terms of flow direction was also reported by measuring the deviation angle at reverse flow conditions in stators and rotors. At the measuring planes in-between the blades, large deviation angles were measured for both stators and rotors. These were in the range [5-65 deg], while the highest

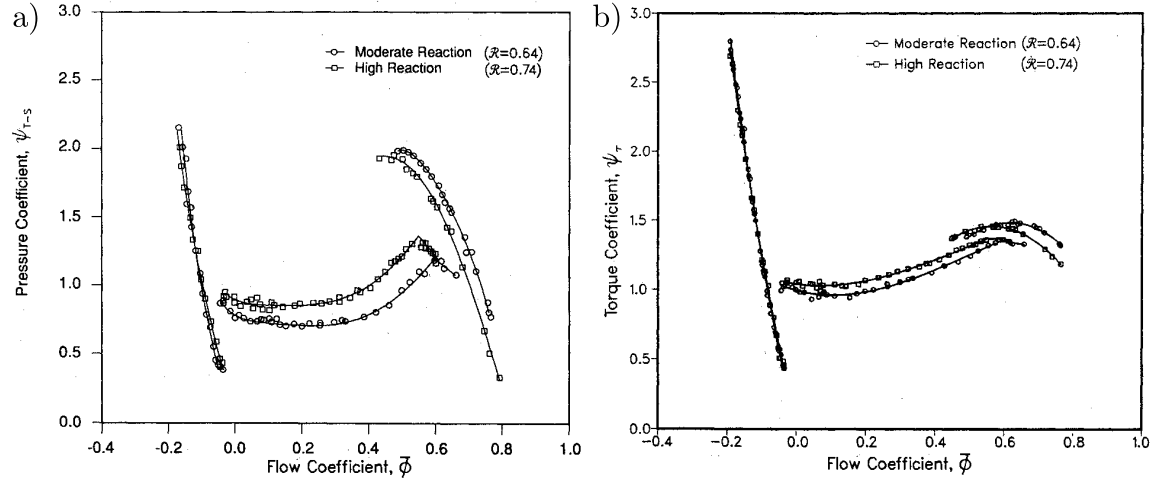


Figure 2.10: a) Total-to-static pressure characteristic and b) torque coefficient characteristic of the MIT 3-stage, axial-compressor rig, including rotating stall and reverse flow characteristics [59].

deviation angles were measured near the hub and the shroud. The jet flow (not separated flow) is well aligned with the camberline until the flow exits the passage, but a significant change in the direction was measured once the flow exited the passage. This was attributed to mixing effects which increased the flow angle. It was also hypothesized that the reason stator blades exhibited larger positive deviation angles was originating to the higher uniformity of the flow downstream the stators, indicating a more mixed-out flow, thereby resulting in larger positive deviation angles with respect to rotor blades (negative deviation indicates that the flow is oriented more towards the axial direction).

In Cambridge university, Day [60] conducted a similar experiment using a 4-stage, low-speed compressor. The compressor was tested at a rotational speed of 3000rpm. The compressor rig was constructed in a modular way allowing the usage of different "internal partitions" and ducts of different lengths, connecting the compressor with the plenum. This allowed the investigation of the compressor's response by varying the volume of the entire system, corresponding to B parameters (eqn.(2.1)) between 0.3 and 1.5. Both classic and deep surge events were captured, while the occurrence of the full-lengthscale instabilities was preceded by rotating stall events. Classic surge was observed when the hysteresis of the cycle was reduced by introducing plates along the exit duct. Day demonstrated that the size of the plenum volume is underpinning, to a large extent but not exclusively, the type of the full-lengthscale instability, i.e rotating stall or surge. Larger plenum volumes resulted in more extended surge cycle trajectories and usually lead to surge events. Reduction in the volume of the system decreased the trajectory of the surge cycle. Once the plenum volume dropped below

a *critical volume*, the compressor was developing rotating stall and surge could not be triggered. The entire compressor map comprising the rotating stall and reverse flow characteristics and the effect of the plenum volume on the surge trajectories are shown in figure(2.11).

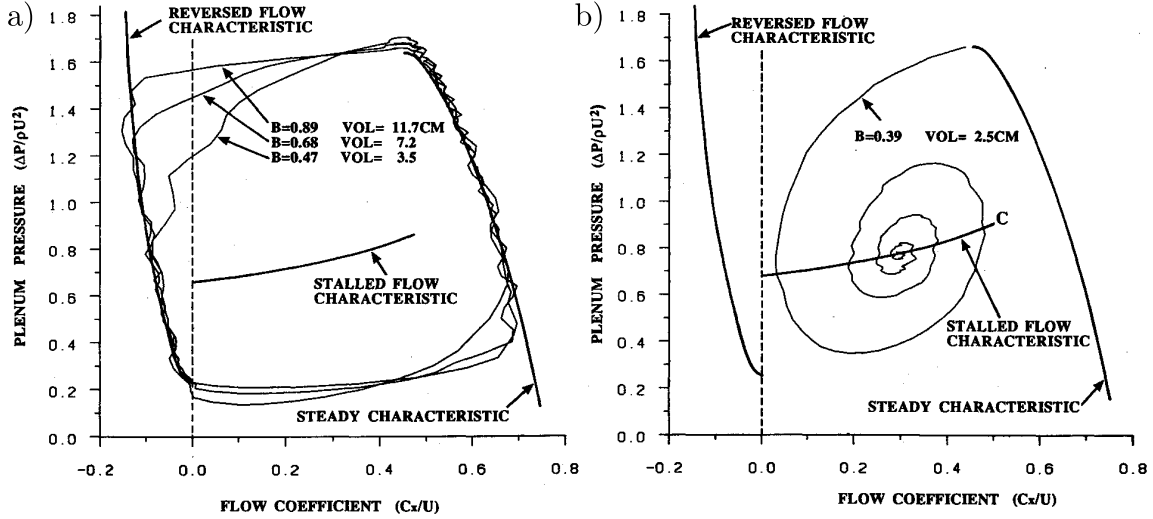


Figure 2.11: a) Deep surge cycle trajectory of a low-speed, 4-stage axial compressor for different plenum volumes and b) the same compressor undergoing rotating stall when the plenum volume is smaller than the critical one [60].

Centrifugal compressors typically undergo more subtle instability phenomena, especially in the rotating stall region (see figure(2.9b)), however are still subject to full-lengthscale instabilities, including deep surge events. In Glenn research centre (NASA), the post-stall aerodynamic behaviour of a centrifugal compressor was experimentally investigated using Digital Particle Imaging Velocimetry (DPIV) methods and dynamic pressure transducers [61, 62]. The compressor is known as NASA CC3 and is essentially an up-scaled version of a Rolls-Royce-Allison compressor, delivering mass flow of 4.54 kg/s at design conditions, compared to 1.66 kg/s of the original design. It generates a pressure ratio of 4:1 at 21789 rpm and is coupled with a wedge-type vaned diffuser and a volute.

Pressure measurements were taken using 2 Kulite sensors in the vaneless space, at different circumferential positions: 1 in the diffuser throat and one upstream the impeller inlet. The existence of a plenum and large duct piping are mentioned but no further details are provided. The compressor mass-flow was adjusted via a collector valve which was gradually closed until the compressor surged and was subsequently opened to allow the compressor to recover. Surge events were successfully captured, with each of the events consisting of 3-5 surge cycles and a frequency of approximately 3 Hz. Rotating stall phenomena were also captured, reporting stall cells rotating at

25%-33% of the rotor speed. At design speed, the diffuser was the first component which stalled. Large pressure oscillations were recorded at the diffuser throat, in the range [170kPa - 540kPa], compared to the 280kPa measured during stable operation. Evidence of reverse flow from the vaned diffuser exiting from the impeller inlet were found but according to the authors, more evidence is required to support that hypothesis.

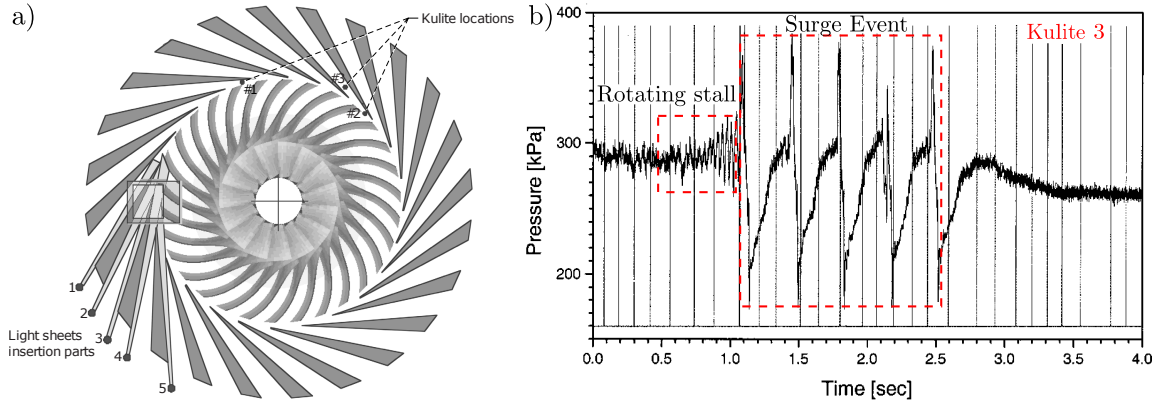


Figure 2.12: a) Overview of NASA CC3 compressor and position of Kulite pressure sensor and b) pressure measurements of Kulite 3 sensor of a surge event preceded by rotating stall [61, 62].

The DPIV measurements elucidated the flow-field behaviour during a surge cycle, focusing on the vaned diffuser throat. Figure(2.13) shows velocity vectors and contours obtained by DPIV measurements at different points during the surge cycle. Figure(2.13a) shows the velocity contours before the surge event, where a low-momentum region downstream the throat can be observed, generating blockage. The impeller is unable to sustain the high-pressure flow in the diffuser and flow reversal is caused, with the flow moving towards the vaneless space (figure(2.13b)). Point (c) corresponds to a pressure similar to the one prior to the surge event; more moderate velocities are recorded while groups of higher and lower velocity are evident, which are presumably related to pressure wave-interaction between the impeller and the vaned diffuser. Point (d) corresponds to the lower pressure ratio of the cycle and represents the point where the impeller reaches equilibrium with the impeller head. The reverse flow velocities in the diffuser are low and forward flow conditions progressively re-establish in the vicinity of the LE. The compressor then gradually recovers and forward flow conditions establish before the cycle is repeated.

At the university of Bath, Powers et al. [63] experimentally investigated the post-stall aerodynamic behaviour of a small turbocharger comprising a small centrifugal compressor. The compressor was coupled with a turbine, powered by a compressed

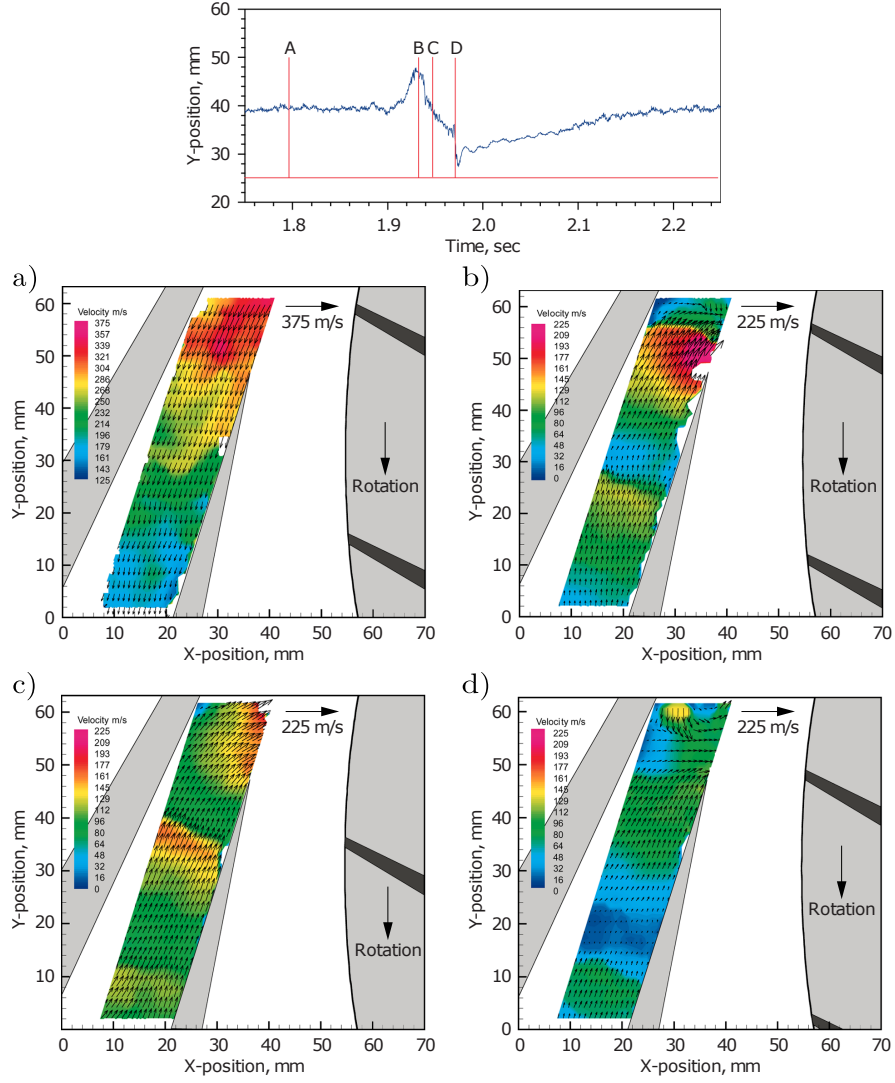


Figure 2.13: Velocity contours and vectors showing the flow-field in the vicinity of the NASA CC3 vaned diffuser TE (95% span) during different phases of a surge cycle [61, 62].

air supply entering at the turbine inlet. A separate mass flow supply was providing air to the compressor in the opposite direction, i.e. air was supplied at the outlet of the compressor during reverse flow to stabilise the operating point. The mass flow of both air streams was adjusted by two separate gate valves. Initially, the mass-flow was supplied only to the turbine, which in turn drove the compressor. Since the compressor valve was closed, the duct was pressurizing and the compressor surged. To stabilise the flow and obtain steady-state operating points along the reverse flow characteristics, the compressor valve was opened to allow the compressed reverse flow stream to stabilise the operating point. A schematic of the experimental configuration along with the total-to-total reverse flow pressure characteristics are shown in figure(2.14).

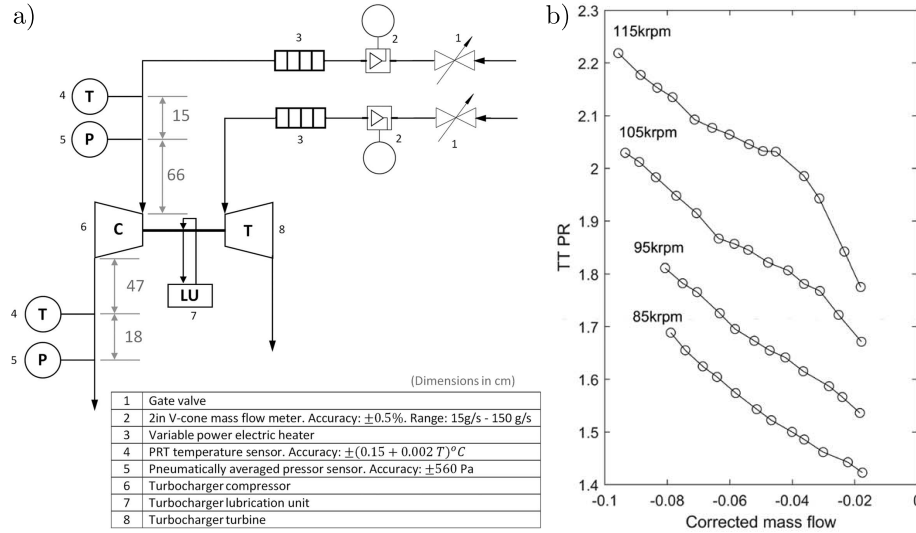


Figure 2.14: a) Schematic overview of the experimental configuration and b) experimental total-to-total pressure reverse flow characteristics of the turbocharger [63].

In the university of Ferrara [32], an Allison multi-stage axi-centrifugal compressor was experimentally tested, aiming to obtain post-stall aerodynamic response data. Two experimental configurations enabled the acquisition of steady-state, forward flow characteristics extending beyond the surge line as well as the emergence and development of full-lengthscale instabilities in the compression system. In the first configuration, downstream the compressor only a short duct with a throttle valve was attached, while in the second, a plenum was added along with an additional duct and a valve to control the mass-flow. The first configuration enabled the establishment of steady, forward flow operating points (forward characteristics) even beyond the surge line, i.e. in the stall region. Surge events were performed with the second configuration. At low rpm, most of the pressure increase was generated by the centrifugal compressor while the axial stages were identified as the destabilizing components of the compressor. Beyond the surge line, the shapes of the characteristics displayed primarily a continuous behaviour, resembling more the behaviour of centrifugal compressors, rather than that of axial ones (figure(2.15)). When operated at low mass flow ratios, deep surge events were triggered, generating backflow. Overall pressure ratio and mass flow measurements along with mass flow and pressure measurements at different locations during a surge event are shown in figure(2.15). At higher rotational speeds, the surge cycle frequency was slightly lower, however the oscillation amplitude of the flow-field variables was larger. Two Kulite pressure sensors were placed at the compressor inlet at 0° and 180° degrees to capture rotating stall phenomena. At 10000 rpm, a stall cell rotating at 13% of the

rotor speed preceding each of the surge cycles was reported.

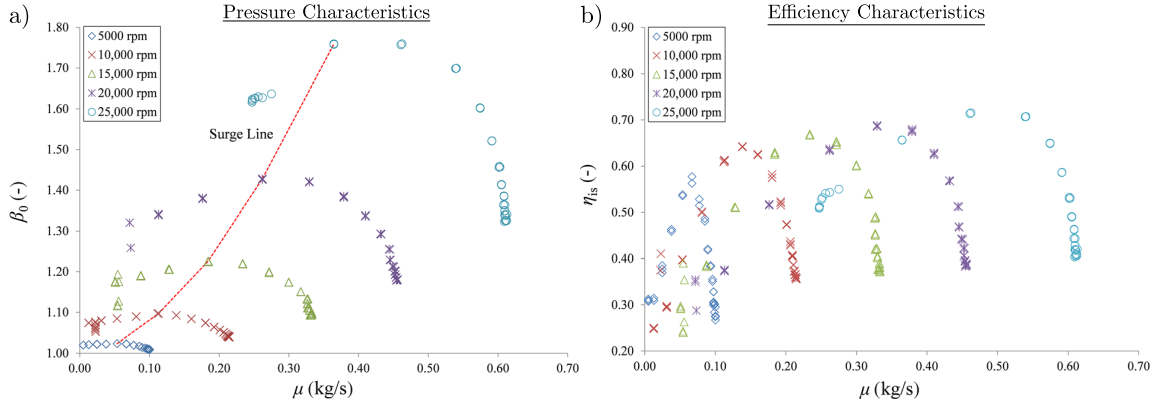


Figure 2.15: a) Pressure ratio and b) efficiency characteristics of the axi-centrifugal compressor extending beyond surge line obtained with first experimental configuration [32].

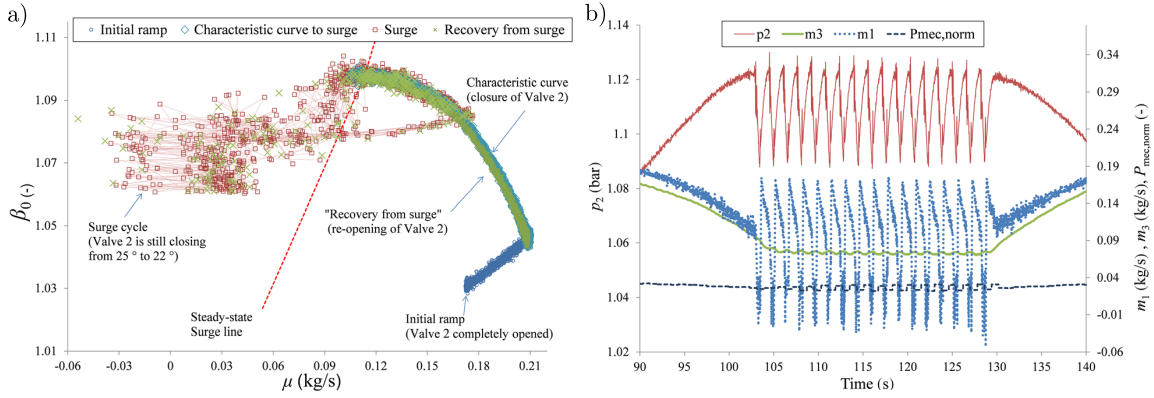


Figure 2.16: a) Pressure ratio and mass flow measurements during deep surge event and b) mass flow and pressure measurements at compressor inlet (1), compressor outlet (2) and downstream plenum (3) obtained with the second experimental configuration [32].

2.3 Compressor aerodynamic performance modelling methods

Compressor aerodynamic performance including pre and post-stall behaviour is quantified by obtaining essential properties at different operating conditions and rotational speeds. These quantities provide essential compressor properties in a concise form over a wide range of operating conditions (compressor maps). The quantities of interest are typically total-to-total or total-to-static- pressure ratio, isentropic or polytropic efficiency, total-to-total temperature ratio (function of efficiency and pressure ratio) and torque. Typically, the aforementioned properties are normalised

and provided in the form of a coefficient while rotational speed and mass flow are corrected using inlet flow properties, i.e. geometric inlet properties at forward flow conditions and geometric outlet properties, at reverse flow conditions. These metrics can be derived using modelling approaches of different levels of fidelity in terms of the lengthscale they resolve the physics. The associated numerical cost and the computational time requirements are intrinsically related to the lengthscale resolution capabilities of the method, thus lower-order methods constitute an efficient alternative approach to conventional 3D CFD simulations. Compressor modelling methods can be categorized as follows; 0D/1D, 2D through-flow and streamline curvature (SLC), 3D through-flow and CFD methods. The latter will not be covered since CFD methods constitute a well known and established method for turbomachinery simulations.

2.3.1 0D/1D methods

Lower-fidelity, 0D/1D methods, are computationally efficient approaches providing rapid performance predictions. They are based on mean-streamline analysis and are entirely relying on inlet and outlet conditions of individual rotors and stators, thus are incapable of providing any insight into the flow-field. The analysis typically relies on empirical correlations or experimental data for deviation/slip and loss estimations. For axial compressors, 0D/1D models usually rely on empirical loss and deviation correlations obtained by 2D experimental cascade data. Aungier's correlations [64] for axial compressors are amongst the most widely used ones and are essentially obtained by curve-fitting experimental data of Lieblein, Howell and Johnsen and Bullock [64]. It is common for manufacturers to develop dedicated correlations during the design process of new compressors which can be used in place of the more generic ones. Loss and deviation correlations for both impellers and axial compressor blades are only available for forward flow conditions, therefore 0D/1D models are generally incapable of predicting compressor post-stall behaviour. An exception is the 0D/1D, mean-line model of Mikhailov et al. [65], which is capable of predicting steady state, reverse flow and rotating stall characteristics. The operating conditions of the compressor were expressed in terms of generalised quantities which were a function of loading and flow coefficients. To enable post-stall predictions, the generalised functions were derived using statistical analysis on a large database of experimental results. The code reproduced the reverse flow and rotating stall characteristics of MIT's low-speed, 3-stage axial compressor rig [59] with reasonable accuracy.

Centrifugal compressor 0D/1D mean-line analysis is widely used during the

preliminary stages of compressor design. Similar to axial compressor design, the analysis is predominantly relying on empirical loss and slip correlations. While in axial compressors, the losses are mainly dissected into profile losses, clearance losses and shock losses [64], centrifugal compressors loss models account for various loss mechanisms e.g. blade loading loss, hub-to-shroud loading loss, incidence loss skin friction loss, mixing loss, clearance loss etc. Some of the most widely used loss models are the one of Oh et al. [66], Galvas [67] and Aungier [68]. Velasquez [69] compared 48 different loss models by combining empirical loss models proposed by different authors. He estimated the efficiency of five different compressor stages using 1D mean-line analysis and compared the results against experimental data. The optimum model was able to predict the efficiencies with less than 1% error while the errors of the least accurate model were approximately 8%.

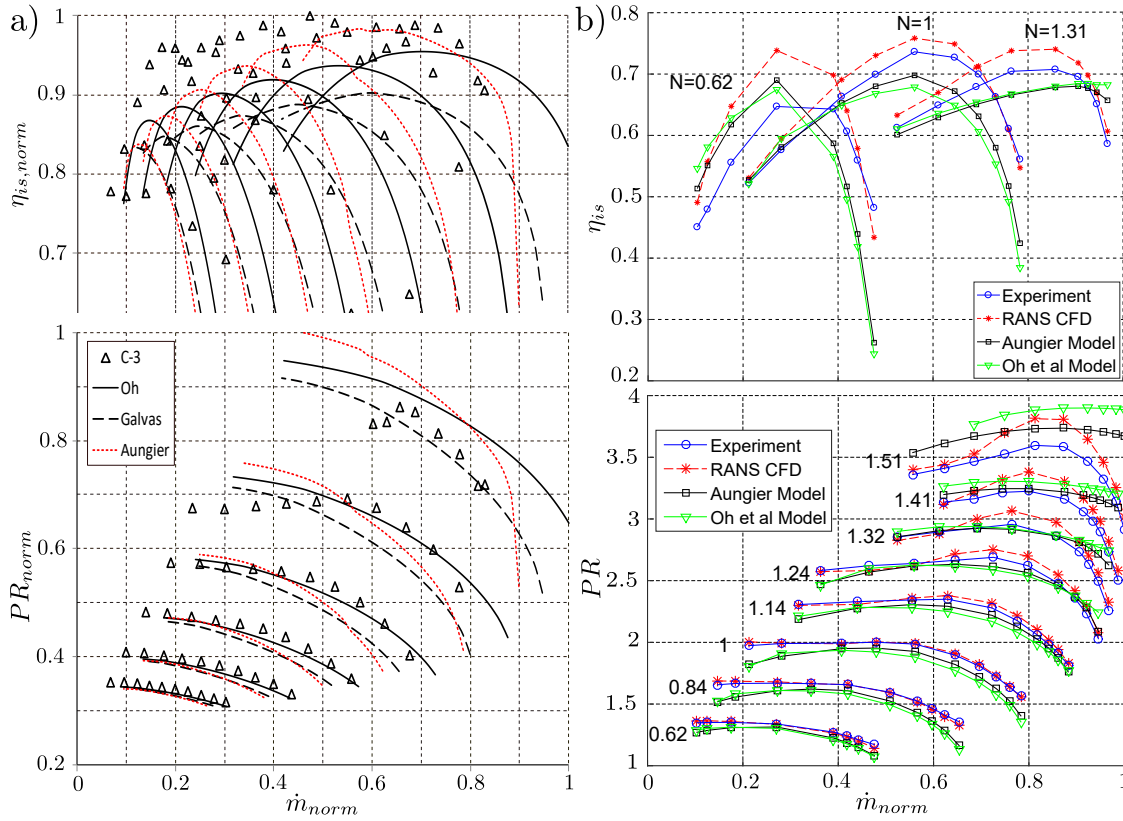


Figure 2.17: Pressure ratio and efficiency predictions comparison using a) mean-line methods and 3 different loss models against experimental data (adapted from [70]) and b) mean-line methods against CFD and experimental data (adapted from [71]).

Kerres et al. [71] obtained and compared performance predictions for a turbocharger for automotive applications using 1D mean-line analysis based on Aungier's and Oh et al. loss model, against CFD and experimental data (figure(2.17b)). Both mean-line models and CFD predictions exhibited larger errors in the higher speed

range with the accuracy of the mean-line predictions being comparable to the CFD ones. The predictions of all models was very accurate in the lower speed range with the 0D/1D models underestimating efficiency and overestimating the pressure rise in the mid to high speedlines. The low order models were substantially less accurate near choking conditions, especially in the higher speed range. Aungier's model was slightly more accurate than the Oh et al. model. The primary sources of errors were identified as the diffuser and the volute. Harley et al. [70] compared the models of Aungier, Oh et al. and Galvas against experimental data of 3 different centrifugal compressor designs for automotive applications. The conclusions regarding the limitations of the lower order models were aligned with Kerres et al. observations; the models are very accurate in the lower speed range but their accuracy diminishes notably in the higher speed range. The predictions become progressively less accurate near stall and choking conditions. The Oh et al. model was concluded to yield better results at lower speeds, but the model of Galvas was substantially better in the higher speed line range while performing reasonably in the lower speed lines, thus was deemed as the best out of the three. A comparison of the three models' predictions for the third compressor configuration is shown in figure(2.17a). No 0D/1D model capable of predicting post-stall aerodynamic performance in centrifugal compressors was found in the literature.

Another modelling approach which also falls in the 0D/1D methods are lumped models. The first analytical lumped-model was developed by Greitzer [72, 73], intending to capture the post-stall transient behaviour of axial-flow compressors. The model comprises an inlet, a constant area-duct, representative of the compressor annulus, an actuator disk which constitutes a mathematical representation of the compressor and a plenum connected with an exit duct and a throttle. The flow in the compressor duct is considered incompressible while in the throttle duct compressible. The pressure in the plenum is considered uniform. The system consists of four non-linear, differential equations expressing momentum conservation in the compressor and the throttle duct, mass-flow continuity in the plenum and a time lag-law to account for the time between the onset of the instability and the development of the full-lengthscale instability (several rotor revolutions). All equations are re-formulated in a non-dimensional form. Experimental pressure characteristics were used as an input for the model. The model was able to successfully predict the transient post-stall behaviour of a 3-stage axial compressor rig, including rotating stall and classic surge events. From that model, the widely known B-parameter was developed, the critical value of which, dictates whether the compressor will surge or stall. It expresses the

ratio of pressure over inertial forces and is given by eqn.(2.1)

$$B = \frac{U}{2\alpha} \sqrt{\frac{V_p}{A_C L_C}} \quad (2.1)$$

An improved model was later introduced by Moore and Greitzer [42, 74], which considered the flow coefficient and the pressure rise coefficient to be not only a function of the axial but also of the circumferential position. They also introduced the concept of axisymmetric characteristics, which are idealised, notional characteristics traversing the entire compressor map (including reverse flow) in the absence of any unsteady phenomena i.e. rotating stall and surge. The model accurately predicted rotating stall and surge and was validated against experimental data [42, 74]. Spakovszky [50] introduced a more complex model, whereby the compression system was represented in a modular way. The model incorporates 5 different modules; 2 modules for the inlet and outlet duct, one for the impeller, one for the vaneless space and one for the vaned diffuser. The vaned diffuser and the impellers are modelled using a semi-actuator disk approach, assuming unsteady, compressible, 1D flow, considering in the analysis pressure losses and neglecting flow deviation. The flow in the ducts is modelled as a 2D flow by recasting the Euler equations using Laplacian transformation and expressing flow variables using Fourier series. Specifying the boundary conditions allows the system of equations to be solved. The eigenvalues of the system then express natural resonances of the system. The model predicted backwards travelling rotating stall waves of the NASA CC3 compressor [62], emanating due to impeller-diffuser interaction. The predictions of the model were subsequently validated experimentally. Powers et al. [75] developed a reduced-order, 1D model by describing the dynamics of the impeller and the diffuser by combining mass, momentum and energy equations into a system of ordinary differential equations. Several assumptions were made to simplify the model e.g. strictly radial flow, straight impeller blades without curvature, vaneless diffuser etc. During reverse flow conditions, a blockage function estimates aerodynamic blockage while skin friction losses are assumed to be only loss mechanism during reverse flow operation. Both models are dependent on coefficients which were experimentally determined. In practice, the skin friction loss accounts for several dissipative phenomena e.g. mixing, separation et. and not strictly skin friction loss. Steady-state characteristics, extending beyond the surge line but not beyond the positive mass flow region were acquired and compared against experimental results of two turbochargers. The accuracy of the model was reasonably good in the lower speed range but larger errors were reported at higher speedlines. A 1D throttle model

was used for dynamic, unsteady simulations and the model demonstrated realistic unsteady behaviour near and beyond the surge line. To deal with fully reverse flow regimes, an improved, revised version of the model was proposed [63]. The model predicted steady-state reverse flow characteristics of a turbocharger with reasonable accuracy at different rotational speeds. The damaged turbocharger and experimental pressure measurements were shown in the introduction (see figure(1.7)).

2.3.2 2D through-flow methods

Marble [76] was the first to introduce the body force concept, adding body force source terms to introduce flow turning and losses. Inside blade rows, the flow-field was considered axisymmetric, thereby assuming that losses and flow-turning are circumferentially uniform, as if the flow was guided by an infinite number of blades. Marble related the body forces to thermodynamic properties and the through-flow concept was first implemented in streamline curvature (SLC) methods. Streamline curvature approaches have been used for decades in the early stages of turbomachinery developments [64]. The solution is derived in the meridional plane by iteratively readjusting the position of the streamlines until equilibrium of continuity, momentum and energy is reached. SLC methods typically encompass empirical correlations to predict losses and flow deviation . They are capable of providing reasonably accurate, steady-state performance predictions including distributions of flow-field variables in the meridional plane. Blade-to-blade, flow-field solution is also feasible and can be combined with the meridional solution. One of the first SLC methods for axial compressors was developed by Denton [77]. In Cranfield, an SLC code for axial compressors called SOCRATES [78], has been developed and validated against experimental results. The streamline curvature method was coupled with a set of loss and deviation correlations and demonstrated its potential of predicting global aerodynamic performance including accurate spanwise profiles of essential flow-field properties in axial-flow compressors. The same code was used to demonstrate the ability of SLC approaches to deal with radial inlet distortions [79]. The fundamental principles underlying SLC methods, are naturally applicable to radial-flow turbomachinery, thus SLC methods can be employed for centrifugal compressor analysis [68, 80]. There are four primary limitations regarding SLC methods: their inability to naturally generate shock waves and consequently choke, their sensitivity to geometric discontinuities, their inability to deal with flow reversal and the axisymmetric flow assumption. The first is typically addressed by introducing shock losses implicitly,

via dedicated loss models such as the one of Aguirre [81] while choking is imposed numerically. Circumferential variations can be considered by combining blade-to-blade and meridional solutions.

Euler-based, through-flow codes resolve the inviscid governing equations along an empty compressor duct, without the presence of distinct discretised blade geometries. Body force-fields are exerted inside bladed domains, imposing flow turning and losses. The body forces are dissected into an isentropic force, acting perpendicular to the flow and imposing flow turning and one parallel but opposite to the direction of motion which generates entropy. At SNECMA, Escuret and Garnier [82] developed an axisymmetric, 2D, through-flow code which solves the unsteady, axisymmetric Euler equations with blade blockage. A 3D version of the code is also reported, solving the 3D unsteady Euler equations in ducted domains, while multiple meridional solutions are obtained in bladed domains. Loss and deviation data were obtained by blade row characteristics. The 2D code predicted the surge point with reasonable accuracy as well as mild-surge behaviour but was unable to cope with fully reverse flow. The 3D version of the code predicted rotating stall at different mass-flows but the accuracy of the rotating stall events were not experimentally validated. Stewart [83] proposed an iterative-based approach for the determination of the body forces to induce a prescribed amount of flow turning and pressure loss and simulated the performance of a whole engine via CFD, RANS methods. Zoppellari [84] used a quasi-2D and full 2D solver to model rotating stall and surge in axial-flow compressors. The quasi-2D version (FENICE-PC) was based on the parallel compressor approach, whereby a stalled compressor is assumed to consist of one or more stalled and unstalled sectors, operating at different operating points at the same time (along the stalled and the unstalled part of the characteristic) [28]. Her model predicted the reverse flow and part of the rotating stall characteristic of MIT's 3-stage low-speed rig [59] with very reasonable accuracy.

Regarding centrifugal compressor 2D through-flow modelling, the only, partially successful body force-based approach incorporating explicit or semi-explicit body force methods (see section(2.4.1)) is the one of Kottapalli (see section(2.4.1)). A semi-explicit modelling approach was used, whereby analytical turning force and viscous force models were combined with CFD-extracted properties. Additional source terms were introduced to the governing equations (Euler) to account for blockage effects. The body force model (eqn.(2.19)), presented in detail in section(2.4.1), entails quantities expressing flow deviation and entropy generation which are obtained by steady-state, single-passage RANS simulations. Body-force simulations were performed in

Fine/Open, by solving the inviscid Euler equations with body force source terms. Due to constraints of the software, CFD-derived quantities and blade metal blockage were expressed using polynomial fits, as functions of the radial and circumferential coordinates (r, ϕ) . The blade metal angle distributions were extracted externally, on a camber surface grid, while the metal blockage distribution was determined in a similar manner using suction side and pressure side data. Results were obtained only for a transonic, 2D, radial-only impeller with vaneless diffuser while numerical instabilities were reported in 3D simulations and no converged solutions were obtained. The global aerodynamic performance predictions were reported in terms of total-to-static pressure rise and stage loading coefficients and the corresponding errors were within 7.75% and 4% with respect to the corresponding RANS estimations. The errors were partially attributed to curve-fitting errors. Higher discrepancies were reported near choke and were linked to absence of aerodynamic blockage and lack of upstream influence in the pressure-field in the vicinity of the LE. The non-uniformity in the pressure field upstream the LE can only be captured when discrete blades are modelled and is not perceived by the body-force approach, resulting in lower radial velocities near at the LE. Similar discrepancies in the choking regions were identified within the present work and were mainly addressed by aerodynamic blockage modelling. A comparison of the RANS and body force pressure rise coefficient predictions along with the change in radial velocity near the TE due to upstream influence introduced by the blades is shown in figure(2.18).

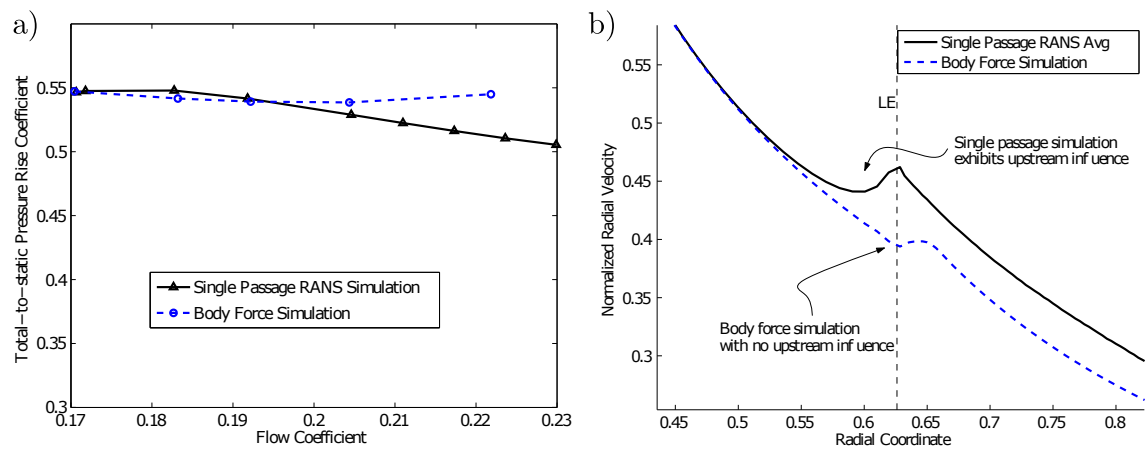


Figure 2.18: a) Comparison between RANS and body force simulations predictions of the pressure rise coefficient of a radial impeller and b) change in radial velocity near LE due to upstream influence introduced by the blades [85].

2.3.3 3D through-flow methods

Gong [86] was the first to demonstrate the capability of body force-based approaches to simulate inlet distortions, stall inception due to short-wavelength perturbations, as well as rotating stall in axial compressors. He developed a 3D Euler-based code and introduced flow turning and viscous dissipation with a simplified, explicit body force scheme. The solver was formulated in the absolute FoR and did not account for blockage effects. Gong's body-force method (see section(2.4.1)) constitutes one of the most important models for turbomachinery simulations but its applicability is restricted to axial-flow machines. Validation of the code was carried out on a 4-stage, GE compressor and a single-stage, axial compressor. Short-wavelength disturbances were triggered by imposing a disturbance of sufficient amplitude near the tip, triggering rotating stall phenomena. The steady-state results were validated against experimental results but no reverse flow could be simulated. Hsiao et al. [87] used Gong's body force model to generate the body force-field of a fan stage comprising the fan rotor and a fan exit guide vane (FEGV) of the NASA 22" rig. The rig consists of a high-bypass ratio nacelle and the aforementioned fan stage. In this approach, the body force coefficients of Gong's model were extracted at every node along the meridional plane (body force-field is considered axisymmetric) via 3D, single passage, RANS simulations. The body-forces were introduced into a 3D, CFD code (WIND) and the engine model was simulated with both discretised blade geometries and body-force representations of the rotor and the FEGV blades. The spanwise profiles of the total pressure, temperature and swirl angle were in good agreement with the experimental data. Flow separation at the intake was also successfully reproduced, with the body-force approach overestimating the separation angle of attack by 2° .

Expanding on Gong's work, several improvements regarding different modelling aspects were subsequently proposed, but were exclusively focused on single or multi-stage, axial-flow compressors. Kiwada [38] introduced the blade-averaging method to more accurately extract body force-fields from CFD flow solutions and added blockage to the inviscid governing equations in the absolute FoR. Brand [88] improved Gong's model by accounting for streamline curvature effects and used Marble's viscous force equation [76] to replicate viscous effects. Essential body-force inputs were extracted from CFD simulations. The pressure rise and work coefficient prediction errors were within 7% and 5% respectively while the adiabatic efficiency errors within 1.5%. Kerner [89] extracted the forces from 3D simulations using control volume analysis instead of blade-averaging methods and successfully reproduced three-dimensional

points in the mid-mass flow region of the characteristic were in good agreement with CFD results but the gradient and the pressure ratio predictions near choke and closer to stall were inaccurate (figure(2.19a)). Stall inception due to both modal waves and spikes was demonstrated using body-force methods on a centrifugal compressor with vaned diffuser (figure(2.19b)). Backwards-travelling modal disturbances were captured at 75% design speed, the presence of which, was recorded experimentally at 100% design speed Spakovszky [50]. No post-stall response i.e. rotating stall or surge could be obtained.

Longley [91] developed a 3D code aiming to simulate pre and post-stall, steady-state and transient aerodynamic performance of multi-stage axial compressors using body force representations of the blade passages. To account for small-lengthscale phenomena with size smaller than a computational cell, Longley modified the inviscid equations by introducing a blockage parameter, b . The blockage parameter expresses the non-uniformity of the flow and the associated momentum and kinetic energy surplus it implies (see section(2.4.1)). By adding a sixth equation to account for the generation and dissipation of that parameter and the associated mixing losses, his modelling approach was capable of accurately predicting rotating stall and surge cycles of multi-stage axial compressors. The code was validated against Gamache's 3-stage compressor rig [59] and Day's C106, 4-stage, low-speed, axial flow compressor rig [60] (figure(2.20a)).

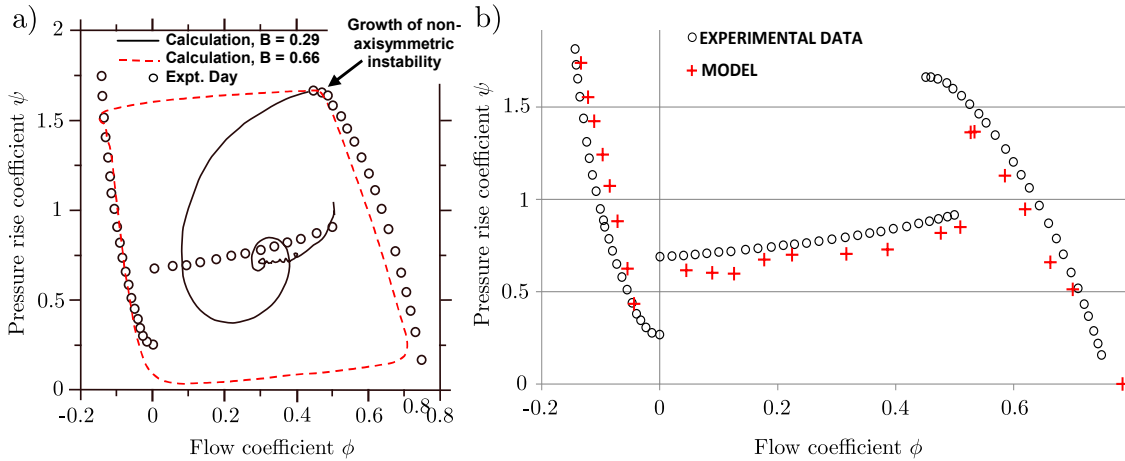


Figure 2.20: Experimental and through-flow predictions of entire compressor map, of the 4-stage C106 rig by a) Longley [91] and b) Righi [27].

Aligned with Longley's ideas, Righi developed a 3D, through-flow code without the additional complexity introduced by Longley's blockage-transport mechanism. He developed a model capable of dealing with forward, reverse and mixed flow regimes inside blade passages and used a combination of Gong's, Longley's and Brand's body

force models to introduce flow turning and entropy generation. Empirical correlations were used at forward flow conditions to avoid dependency of the code on CFD inputs. The proposed modelling approach was capable of accurately simulating rotating stall and surge events in multi-stage, axial-flow compressors [27, 92] and was validated against Gamache's [59] and C106 rig [60] (figure(2.20b)). Further validation of the code was later performed against 3D CFD, RANS data of a modern, six-stage, high-speed, high-pressure, axial-flow compressor. More recently, the post-stall behaviour of a high-speed, 8-stage, modern axial-flow compressor was investigated and compared against high-fidelity, 3D uRANS simulations performed at Imperial college using the in-house RANS code AU3D. Pressure rise, mass flow and temperature traces during the simulated surge events were in good agreement with the CFD ones. The evolution of rotating stall cells emerging before the surge event and during the recovery phase were successfully captured by ACroSS, predicting similar rotating stall patterns [58] (figure(2.21)).

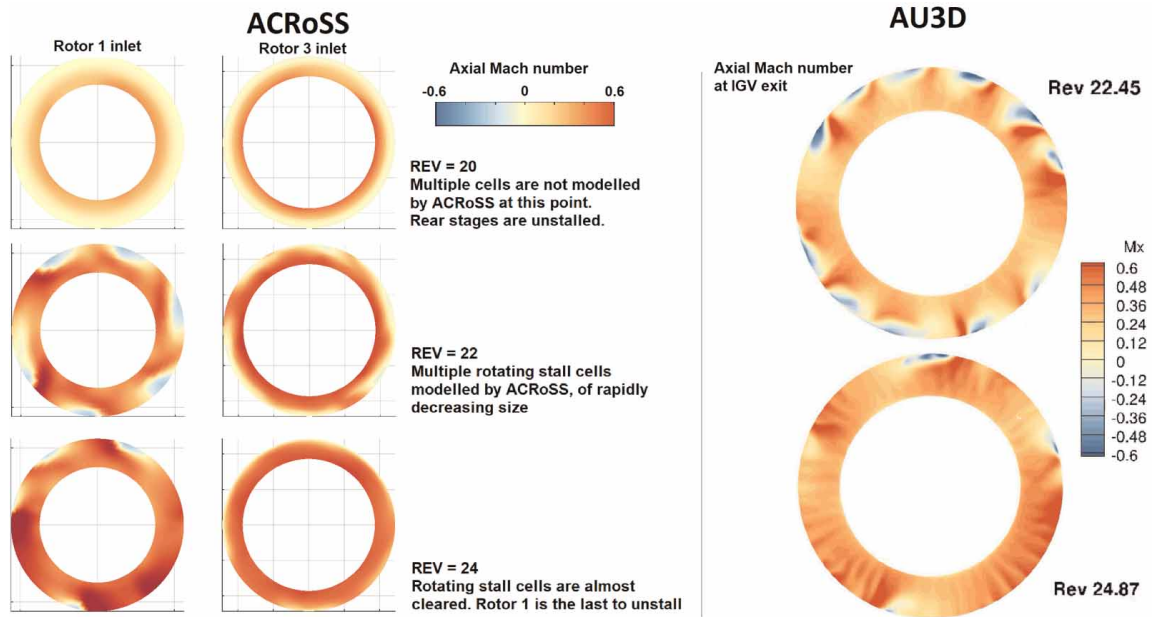


Figure 2.21: Axial Mach number contours showing the evolution of rotating stall cell patterns at different rotors and shaft revolutions, predicted by ACroSS and uRANS simulations [58].

The code was further validated [93] against experimental data during the Rig250-build 6B test campaign in DLR institute of propulsion and technology. The rig consisted of 4.5 stages and is representative of a modern engine HPC system, while the outlet of the compressor was connected to a variable-volume plenum. Surge cycle data were obtained at design speed for 2 plenum volumes. Unsteady flow-field measurements were successfully reproduced, by performing axisymmetric (2D),

through-flow simulations. The study demonstrated the significance of the inlet bellmouth, plenum and exhaust piping in the dynamic response of the compression system during surge events. The length of the inlet duct connecting the compressor with the settling chamber was shown to substantially affect the period and the behaviour of the surge cycle. Inclusion of those components in the numerical models was deemed necessary to accurately capture the surge response of the compressor.

2.4 Lower-order modelling methods

Conventional CFD turbomachinery methods require modelling and discretisation of the duct and blade geometry. Depending on the intended application, the numerical solution is derived either by single-passage simulations, or full annulus simulations, if the presence of circumferentially non-uniform phenomena of lengthscale higher than a blade pitch are expected. Grid refinement requirements need to be met near wall surfaces to properly resolve shear stresses and boundary layer effects, further increasing the overall computational cost. The computational cost is dramatically exacerbated in transient simulations e.g. rotating stall or surge simulations. Through-flow methods typically incorporate inviscid (Euler) solvers coupled with body-forces to introduce flow turning effects in conjunction with other, lower-order methods to estimate quantities such as flow deviation, pressure loss, aerodynamic blockage etc. These lower-order methods are usually simplified analytical models, empirical correlations or datasets derived from CFD simulations or experimental data. The inclusion of lower-order models is imperative since these phenomena are predominantly related to dissipative phenomena and are therefore not intrinsically replicated by inviscid solvers. This enables the reproduction of viscous-related phenomena implicitly, making the through-flow approach more accurate while drastically reducing the computational cost requirements, since no distinct, discretised blade geometries and refined grids are required.

2.4.1 Body force methods

The body force models typically consist of two primary components: the *turning force* and the *viscous force*. The first introduces flow turning, i.e. changes the direction of the flow isentropically and is therefore acting perpendicular to the relative flow path. The second is acting parallel to the relative streamline but in the direction opposite to the one of motion, thereby generating entropy [91, 94]. The body force

methods can be classified in 4 main categories based on the inputs they require and the way they are extracted: explicit, semi-explicit, CFD-extracted database methods and Immersed boundary methods with smeared geometry [94].

Immersed boundary methods with smeared geometry (IBMSG)

Immersed Boundary Methods (IBM) are methods which account for wall boundaries in simple grids without the grid being aligned with the actual modelled geometry [95]. Inside each cell, information about the wall boundary is interpolated and the flow is turned via a body force field so that the wall boundary condition is satisfied. This concept was successfully applied in the past in simulations of complex 3D flows with moving boundaries [95]. Cao et al. [96] expanded that concept by developing the so called Immersed Boundary Method on Smeared Geometry (IBMSG) and demonstrated its ability of predicting the steady-state performance of a fan rotor (NASA rotor 67) as well its behaviour at 10% inlet distortion. A simple, uniform grid was used, with each cell positioned inside the bladed domain containing information about the local wall boundaries. The actual camberline of the blade is essentially approximated by a smeared one, composed by the small blade fractions contained in each cell (figure(2.22)). The flow turning source terms are computed at every timestep by solving the momentum equations for the turning force component in the direction normal to the local blade wall (fraction of camberline) using forward time-steppng discretisation and applying wall boundary conditions so that the velocity normal to the blade wall is zero.

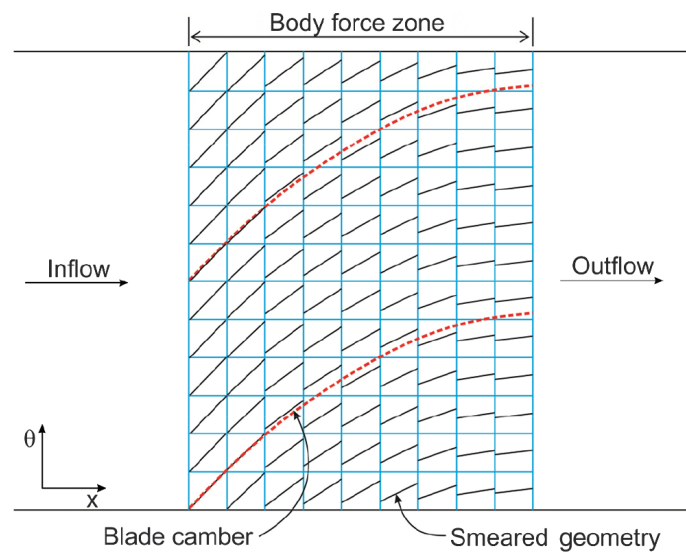


Figure 2.22: Illustration of the smeared camberline representation of the IBMSG model [96]).

The turning force is computed by solving the discretised (forward time stepping) momentum equation in the direction normal to the wall for the turning force component, thus precisely estimating the amount of force required to align the flow with the wall boundary:

$$f_{turn} = -\frac{\mathbf{W} \cdot \hat{\mathbf{n}}}{\Delta t} - \mathbf{RHS} \cdot \hat{\mathbf{n}} \quad (2.2)$$

In the above equation, **RHS** indicates all the terms in the right hand side of the momentum equation. The viscous force model of Cao et al. contains the contribution of three components; surface frictional loss, secondary loss due to negative pressure gradients, and endwall loss. All components are scaled using coefficients, with K_{p1} and K_{p2} being constant coefficients for the first two components while C_d is the drag coefficient. These coefficients are presumably tuned for each blade geometry to match experimental data. The viscous force model along with its constituent components are given below:

$$f_{loss} = f_{pb1} + f_{pb2} + f_{pb3} \quad (2.3)$$

$$f_{pb1} = -K_{p1}\rho\mathbf{W}\mathbf{W} \quad (2.4)$$

$$f_{pb2} = -K_{p1}\rho\frac{\partial P}{\partial x}\frac{U_{tip}^2}{u_x^2}\frac{\mathbf{W}}{u_x} \quad (2.5)$$

$$f_{pb3} = -C_d\frac{1}{2}\rho|\mathbf{W}|\mathbf{W}A \quad (2.6)$$

,where u is the axial velocity, $\partial p/\partial x$ the axial pressure gradient, U_{tip} the blade tip speed and A the cell surface area on the endwalls.

CFD-extracted database methods

Another approach of estimating the turning and the viscous forces is implicitly, via CFD simulations. The blade is initially modelled using a regular CFD approach and the flow-field solution is typically obtained by means of steady-state, 3D, single-passage, RANS simulations. The flow-field solution is subsequently circumferentially averaged and the forces are extracted from an axisymmetric flow-field. Kiwada [38] introduced a new averaging method whereby the averaged flow-field is a blade-average of the original 3D flow-field, thus accounting for circumferential variations of the flow

properties. Body forces were extracted in a range of operating points, including points beyond the stability limit of an axial compressor stage with an inlet guide vane. The pressure rise and the separation in the hub was predicted with reasonable accuracy. Using a blade-averaged flow-field, the momentum equations can be solved to obtain the body force components in the three directions as per eqn.(2.7) [49]. It should be noted that the extraction of the body force source terms does not dissect the turning from the viscous force; a single component is obtained in the axial, tangential and circumferential component accounting for the combined contribution of both.

$$\mathbf{f} = \frac{\iint_S (\rho \mathbf{V} \cdot d\mathbf{S}) \mathbf{V} + \iint_S P \cdot d\mathbf{S}}{\iiint_V \rho dV} \quad (2.7)$$

Benneke [49] followed a similar approach, adopting the circumferential averaging method of Kiwada [38] and applied it to a centrifugal compressor with vaned diffuser. He generated a large data base, a "look-up table", by performing CFD simulations corresponding to 53 operating conditions and 3 speedlines by changing the boundary conditions at each rotational speed. Normalised axial, tangential and radial force components were extracted as a function of the local Mach number and swirl angle. The data were used to generate a response surface model based on Gaussian process regression with Kriging scheme, to obtain interpolated and extrapolated values for a range of different conditions, using the initial CFD dataset as a basis (figure(2.23a)).

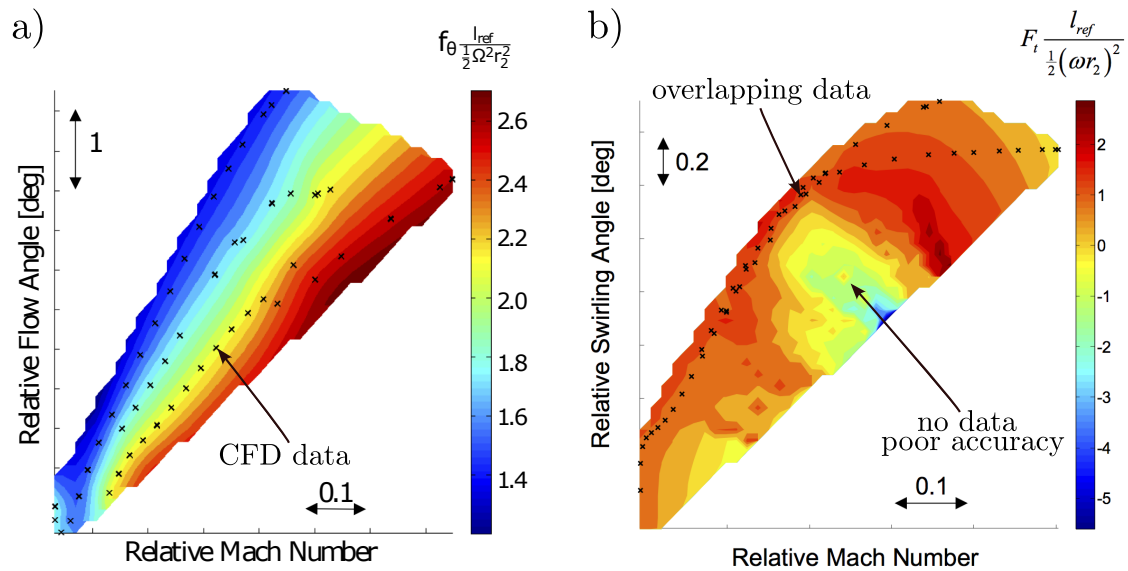


Figure 2.23: a) Benneke's surface model of tangential body force component as a function of relative Mach number and relative flow angle and b) data corresponding to 3 speedlines, collapsing into a single curve, compromising the accuracy of the response surface model [49].

This approach was readjusted to a simpler, 1D approach, whereby the body forces were given exclusively as a function of the meridional Mach number. The simplified approach was eventually adopted, since in several occasions, the datasets of all speedlines would collapse into a single curve, thus not providing enough information in the direction normal to the curve (figure(2.23b)). This impeded the acquisition of reliable approximate values from the response surface, introducing large errors and instabilities. Patel [97] used Denton's streamline and 3D RANS code (TBLOCK), to extract body forces following Benneke's approach and proposed a method for joining the body forces extracted from the two methods. To overcome the limitation of Benneke's approach, he linked the body forces to the local flow coefficient and the overall flow coefficient.

Explicit models

Explicit body force models are analytical or semi-analytical, physics-based models of the forces exerted on the flow due to the presence of blades. These models are typically functions of various flow-field variables thereby naturally readjusting to changes in flow variations without relying on extensive CFD-derived databases. The first, physics-based model for axial compressors was developed by Gong, intending to capture short-wavelength stall phenomena, rotating stall events and simulate compressor response to three-dimensional inlet distortions [86, 98]. Gong's model adopts a simplified representation of the blade passage, considering the blade passage as a series of staggered flat plates without curvature (figure(2.24)). The model assumes two-dimensional flow (no radial components), omitting the presence of radial forces and blade leaning. Since the blade is assumed a staggered straight passage, a pressure gradient is only acting in the direction parallel to the passage, while the pressure gradient normal to it is zero. This constitutes the base component of the turning force model, referred to as blade loading force. The second turning force component is the flow-tangency force; this essentially is a correction force and acts whenever there is a misalignment between the imposed direction and the actual flow direction. Its magnitude is automatically readjusted by the extent of the local misalignment and an empirical coefficient K_n . The local lengthscale used to obtain the correct units is the local blade pitch. The pitch was preferred over the chord which was deemed a more appropriate lengthscale, because it can be defined locally [85]. The viscous force model is analogous to the drag of an aerofoil, adopting the blade pitch as the lengthscale while an empirical constant similar to the drag coefficient of an aerofoil is used to scale the viscous force. These coefficients were determined using loss and

deviation correlations at midspan and blade tip locations, with K_n being a function of the blade metal angle and K_p a constant [88]. Gong's turning force model, expressed as a function of the blade loading and the flow tangency force, is given by eqn.(2.8):

$$f_{turn} = \frac{\overbrace{1 \frac{\partial P}{\partial x} \sin \kappa}^{F_{n,blade \ loading}} + \frac{\overbrace{w_\eta w_\xi}^{F_{flow \ tangency}}}{K_n s} \quad (2.8)$$

The viscous force component is estimated by eqn.(2.9).

$$f_{visc} = K_p \frac{w^2}{s} \quad (2.9)$$

,where η and ξ are the directions parallel and perpendicular to the blade passage respectively and s the local blade pitch. Peters [99] later modified the turning force model of Gong by accounting for blade lean; this introduced a radial force component which was neglected by Gong's original model. A schematic overview of Gong's model, the coordinate systems and the simplified blade passage is shown in figure(2.24).

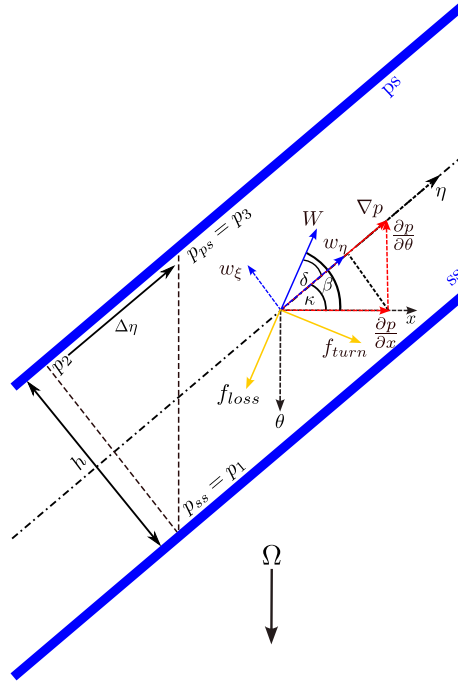


Figure 2.24: Representation of Gong's blade passage model and body force components (adapted from [99]) .

Another body-force model which falls in the category of explicit body force models, is the one of Longley [91]. It was used in a custom-developed, 3D, through-flow code to predict the entire compressor map of a multi-stage axial compressor including forward, reverse and rotating stall characteristics. The circumferential component of

Longley's turning force model in steady-state conditions is given by eqn.(2.10). During unsteady conditions, a time lag law is used to account for shed bound circulation which convects downstream the passage affecting the body forces. The force in the meridional direction is then given as a function of the circumferential component and the relative flow angle (eqn.(2.11)). The over-line superscript indicates averaged conditions, but testing of the model within the current project confirms that its compatibility is not restricted to averaged quantities.

$$f_{turn\theta}^{steady} = \overbrace{\rho \bar{V}_m^2 \frac{1}{R_{curv}}}^{F_{blade\ loading}} + \overbrace{\rho |\bar{V}_m| (\bar{w}_{theta} - \bar{V}_m \tan \kappa) \frac{K}{s \cos \kappa}}^{F_{flow\ tangency}} \quad (2.10)$$

$$f_{turn\theta}^{steady} = -f_{n,turn\theta} \tan \beta \quad (2.11)$$

Similar to Gong's model, Longley's turning force model (eqn.(2.10)) can essentially be dissected into two constituent components. The first component is the blade loading component or the base turning force component, providing the primary turning of the flow. The R_{curv} term, represents the radius of the camberline curvature, thus considering blade curvature effects which were neglected by Gong's staggered, flat plate representation of the blade passage. The second term is similar to the flow tangency model of Gong and acts whenever the flow is not aligned with the imposed direction. The coefficient, K , is in the range [0.4 - 2.0] at design conditions. The viscous force is approximated by eqn.(2.12) and is a function of the total pressure loss, the static and total pressure as well as the chord, c . The deviation and pressure loss at forward flow conditions are predicted using Howell's empirical correlations.

$$f_{loss} = -(\Delta P_0) \frac{P/P_0}{c} \quad (2.12)$$

Righi [27, 92] adopted a similar body-force approach to simulate the pre and post-stall aerodynamic behaviour of multi-stage axial compressors. He merged the base formulation of Brand [88], substituting the meridional flow angle gradient term with the radius of the blade's curvature (similar to Longley) and added Gong's flow tangency force (referred to by Righi as correction force) to ensure control of the flow when high gradients are encountered. At forward flow conditions, pressure losses and deviation are predicted by a set of empirical correlations proposed by Aungier [64]. The correlation of Thomason and Moses [100] for separated flow is used when the flow is stalled. The base-turning force component and correction force proposed by Righi are given by eqn.(2.13) and eqn.(2.14) respectively.

$$f_{turn} = \frac{1 - Ma_x^2}{\cos^2(\beta)(1 - Ma_{rel}^2)} \cdot \frac{\rho u_\eta^2}{R_{curv}} + \frac{\tan\beta(1 + (\gamma - 1)Ma_{rel}^2)}{1 - Ma_{rel}^2} f_{visc} + f_{cor} \quad (2.13)$$

$$f_{cor} = -\rho |u_x| \frac{u_\xi}{s \cdot \cos\beta} \cdot K_{cor} \quad (2.14)$$

In the above equations, R_{curv} is the radius of the curvature corresponding to the imposed relative flow path, i.e. local blade camberline if no flow deviation is considered, β the relative flow angle, s the local blade pitch, u_{xi} the velocity component perpendicular to the imposed flow-path and u_{eta} the velocity component parallel to the imposed flow path. The viscous force is identical to Longley's but the chord is substituted by the local flow-path length and is given by eqn.(2.15)

$$f_{visc} = -\Delta P_0 \frac{P/P_0}{\Delta x \cdot \cos\beta} \quad (2.15)$$

Hall [101] developed an incompressible flow-turning model which is not dependent on CFD inputs or calibration coefficients. The model is a function of the local momentum, the local deviation angle (δ) with respect to the local camber surface, the local blade pitch, s and the magnitude of the tangential projection of the normal to the local camber surface vector, $|n_\theta|$. As the solidity approaches zero, ($s/c \rightarrow 0$), the flow is perfectly aligned with the local camber line, i.e. $\delta = 0$, while the model approximates the theoretical lift coefficient of a flat plate aerofoil $2\pi\delta$, as solidity approaches infinity, ($s/c \rightarrow \infty$) [94, 101]. The turning force component of Halls' model is approximated by eqn.(2.16). Thollet later [94] introduced a compressibility correction using the Prandtl-Glauert compressibility correction for subsonic conditions and the Ackeret correction for supersonic Mach numbers.

$$f_{turn} = 2\pi\delta W^2 \frac{1}{s} \frac{1}{|\hat{n}_\theta|} \quad (2.16)$$

Semi-explicit models

This category comprises body-force methods which incorporate explicit, physics-based models coupled with a database which provides necessary inputs for those models. The additional inputs are usually derived by means of steady-state RANS simulations and expanded using response surface models to cover a wider range of conditions. Brand [88] improved Gong's body force model by accounting for streamline curvature which in Gong's model was neglected, since the blade passage

was considered to consist of a series of staggered flat plates. A physically erroneous behaviour of the model is the assumption of zero turning force when the local blade angle approaches zero; a turning force is still required to balance out the centripetal acceleration introduced due to the curvature of the blade. This inconsistency was nevertheless partially counteracted by the flow tangency force.

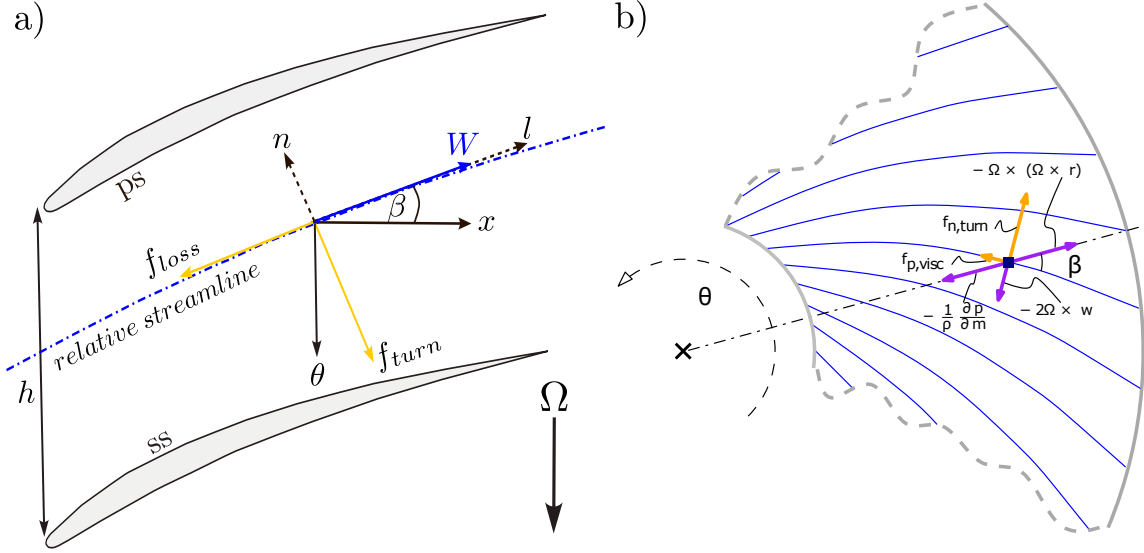


Figure 2.25: Overview of a) Brand's body force model [88] and b) Kottapalli's body force model [85].

To account for blade and consequently, streamline curvature effects, Brand [88] derived a more sophisticated and physically accurate turning force model. Using momentum balance normal to a relative streamline, assuming axial, axisymmetric flow and neglecting blade lean effects as well as Coriolis and centrifugal accelerations, he derived an improved turning force model which addressed the limitations of Gong's model (figure(2.25a)). The axisymmetry of the body force implies the turning force is circumferentially uniform, as if an infinite number of blades were guiding the flow. Brand's turning force is expressed by eqn.(2.17).

$$f_{turn} = \frac{-w^2(1 - Ma_x^2)}{\cos^2(\beta)(1 - Ma_{rel}^2) + K_{off}} \cdot \left(\frac{\partial \kappa}{\partial x} \cos \beta + \frac{1}{c} \Delta \right) + \frac{\tan \beta (1 + (\gamma - 1) Ma_{rel}^2)}{1 - Ma_{rel}^2} f_{visc} \quad (2.17)$$

In eqn.(2.17), $\Delta = c \partial \delta / \partial l$ is a non-dimensional quantity expressing the streamwise deviation gradient. It is interesting to note, that the initial formulation of the turning force was a function of the axial pressure gradient, $\partial P / \partial x$. This was reported to cause instabilities generating a positive feedback with the normal force. An influence coefficient analysis [102] was performed and the pressure gradient term was expressed

as a function of other flow variables, eliminating the aforementioned instabilities. The viscous force model is based on Marble's expression of the viscous force which relates the viscous force with the meridional entropy gradient, the temperature and the local flow angle as per eqn.(2.18).

$$f_{loss} = -T \frac{\partial s}{\partial m} \cos \beta \quad (2.18)$$

Expanding on Brand's ideas, Kottapalli [85] derived a more sophisticated model, applicable to centrifugal compressors. The model was derived by taking momentum balance normal to a relative streamline while considering Coriolis and centrifugal forces (figure(2.25b)). In impellers, the magnitude of those forces is comparable to inertial and pressure forces due the substantial changes in the mean-streamline radius. The model also accounts for blade leaning. Similar to Brand's model, Kottapalli's model was reformulated to eliminate its dependence on the meridional pressure gradient which triggered numerical instabilities. The final form of the turning force model is given by eqn.(2.19).

$$\begin{aligned} f_{turn} = & \left\{ \left[\frac{1}{1 - \frac{\sin^2 \beta (1 + \sin^2 \lambda (\gamma - 1) Ma_m^2)}{1 - Ma_m^2}} \right] \left\{ w^2 \left[\frac{\partial \kappa}{\partial m} \cos \beta - \sin \lambda \frac{\partial \phi}{\partial m} \cos \beta + \frac{\sin \beta \cos^2 \lambda \sin \phi}{r} \right] \right\} \right. \\ & + 2\Omega w [\cos \lambda \sin \phi] + \Omega^2 r [\cos \beta \sin \lambda \cos \phi + \sin \beta \sin \phi] + \frac{u^2}{r} \left[\frac{-\cos \phi \sin \lambda \cos \beta - \sin \phi \sin \beta}{1 - Ma_m^2} \right] \\ & \left. + f_{visc} \left[\frac{-\cos \beta \sin \beta (1 + (\gamma - 1) Ma_{rel}^2 (1 - \sin^2 \lambda \sin^2 \beta))}{1 - Ma_m^2} \right] + w^2 \frac{D}{r_2} \right\} \quad (2.19) \end{aligned}$$

In the above equation, κ is the blade angle, λ is the lean angle, ϕ the flow path angle, i.e angle of the duct in the meridional plane, r_2 the impeller tip radius and the $D = r_2 \partial \delta / \partial r_2$ the normalised streamwise deviation gradient. Kottapalli used a similar viscous force model to that of Gong, expressing the viscous force as a function of the impeller tip radius, the relative velocity magnitude and a coefficient, K_p , which is analogous to an aerofoil drag coefficient (eq.(2.20)).

$$f_{visc} = K_p \frac{w^2}{r_2} \quad (2.20)$$

The body force models of Brand and Kottapalli, despite being derived analytically using fundamental streamline curvature concepts, entail quantities which are not a

priori known. These quantities are the meridional entropy gradient or K_p coefficient and the normalised streamwise deviation gradient, Δ . In their studies, the quantities were extracted by means of 3D, single-passage, steady-state simulations at different operating conditions and were subsequently introduced as source terms in a commercial solver. Implementation of eqn.(2.18) is applicable strictly along streamlines. Radial shift of the streamlines, for example near the hub or the tip, causes mismatch of the pitchwise averaged streamlines, grid-streamlines and the actual streamlines. If the gradients are computed along the grid-streamlines, locally negative streamwise gradients may be computed. Instead, the entropy gradient was assumed to linearly vary between the LE and TE and was computed using LE and TE entropies at different spanwise positions (defined by the blade grid). It should be noted that neither Brand's nor Kottapalli's models include a correction force and to the author's experience, this is a severe limitation for accurate, transient modelling.

A variation of Gong's and effectively Kottapalli's viscous force model, is the one of Peters [99]. The limitation of Gong's loss component is reflected on its inability to properly reproduce off-design losses, especially near choke and stall, which in axial compressors manifest in the form of a loss bucket. This was addressed in Peter's viscous force by introducing a quadratic dependence of the stagnation pressure loss with respect to the relative Mach number. The improved viscous force model of Peters is given by eqn.(2.21).

$$f_{loss} = \frac{K_{p1}}{h} \left[Ma_{rel}^2 + K_{p2}(Ma_{rel} - Ma_{ref})^2 \right] W^2 \quad (2.21)$$

The loss model of Peters is classified as a semi-explicit scheme since it relies on CFD input to determine the K_{p1} , K_{p2} coefficients and the reference Mach number, Ma_{ref} . The reference Mach number is the relative Mach number at peak efficiency, determined by RANS simulations, while K_{p1} , K_{p2} are determined in an iterative manner by comparing body force simulations against single-passage, CFD results.

Another similar model which falls in the same body-force category to that of Brand and Kottapalli, is Sorensen's model [103]. He developed the first body-force model adequate for cavitating inducer simulations in rocket engines. His model is formulated following a similar approach to that of Kottapalli but omits lean angle effects and assumes incompressible flow. However, Sorensen introduced in his model an additional force component, referred to as *binormal force*. That force is exerted in a direction perpendicular to the plane formed by the turning and the viscous force. The inclusion of the binormal force component was deemed necessary to capture the

strong radial flow phenomena encountered in inducer cavitation flows. Source terms were also added to replicate the effects of cavitation blockage. CFD simulations were performed to extract the required input for the analytical body force models achieving encouraging results. Further details will not be provided since the model does not lie within the scope of the present work.

2.4.2 Stalled flow modelling methods

Highly non-uniform flows, similar to the ones encountered at near-stall and post-stall conditions are normally characterised by complex, viscous related phenomena such as vortex shedding, flow separation, mixing etc. Modelling of those complex phenomena is particularly challenging using through-flow methods since these are usually short-lengthscale phenomena, relative to the blade pitch size. In through-flow modelling approaches, the size of the cells, in the circumferential direction typically represents lengthscales corresponding to more than a blade pitch, making direct modelling of those phenomena infeasible.

Longley's model

According to Longley [91], there are three important aspects associated with short-lengthscale phenomena: the increased momentum of the non-uniform flow compared to the momentum of the uniform one, the additional kinetic energy due to the higher momentum of the non-uniform flow and the entropy generation due to mixing-out of those non-uniformities. He proposed a blockage parameter which is expressed as the ratio of the momentum of the non-uniform flow, over the momentum of a uniform one. While the average velocity (for instance volume or area-averaged) of a uniform and non-uniform flow is identical (eqn.(2.22)), the averaged square velocities and consequently the momentum and kinetic energy of the non-uniform flow is always higher than that of the uniform flow. The blockage parameter is therefore always larger or equal to 1, i.e. $b \geq 1$ as shown by eqn.(2.23).

$$\frac{\int \rho u dV}{\int dV} = \rho \bar{u} \quad (2.22)$$

$$b = \frac{\int \rho u^2 dV / \int dV}{\rho \bar{u}^2} \geq 1 \quad (2.23)$$

The excess kinetic energy associated with the higher momentum of the non-uniform flow is computed by introducing a sixth equation in the governing equations,

called the *blockage-transport equation* which dictates the change in the excess kinetic energy due to body-forces and mixing-out of the short-wavelength non-uniformities (eqn.(2.24)).

$$\rho T \frac{Ds}{Dt} = \left(\frac{b|\bar{\mathbf{u}}|}{1/2 s} \right) \frac{1}{2} \rho (b-1) \bar{\mathbf{u}}^2 \nabla \cdot (\rho \bar{\mathbf{u}}) + \left(\frac{b-1}{b} \right) (\rho \bar{\mathbf{u}} \cdot \nabla) \left(\frac{1}{2} b^2 \bar{\mathbf{u}}^2 \right) = -\bar{\mathbf{u}} \cdot \mathbf{F} - \rho T \frac{DS}{DT} \quad (2.24)$$

The entropy production due to blockage-mixing and the mixing rate of the short lengthscale non-uniformities are computed by eqn.(2.25). Complete flow mixing is assumed to take place in a distance equal to the time the jet flow requires to cover a distance equal to half a blade pitch.

$$\rho T \frac{Ds}{Dt} = \left(\frac{b|\bar{\mathbf{u}}|}{1/2 s} \right) \frac{1}{2} \rho (b-1) \bar{\mathbf{u}}^2 \quad (2.25)$$

While the aforementioned method constitutes a novel, highly sophisticated numerical approach for stalled flow modelling, its implementation is challenging. In the same publication, Longley [91] presents a simplified, analytical model which estimates the aerodynamic blockage at reverse flow conditions as well as the associated losses due to mixing-out of the jet flow and the separated flow. This formed the basis of the improved blockage-mixing model presented later in the methodology section. A schematic overview of Longley's simplified blockage-mixing model is shown in figure(2.26).

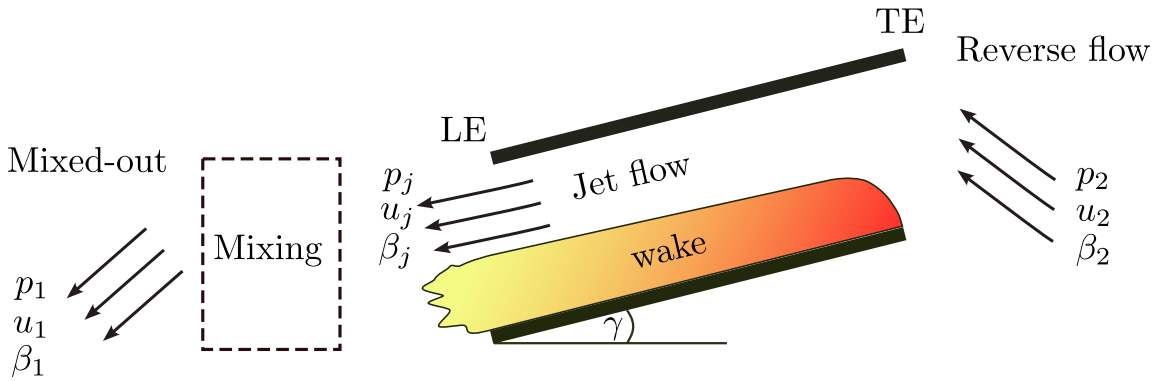


Figure 2.26: Schematic overview of Longley's simplified reverse flow model (adapted from[91]).

The model is derived based on the following assumptions: blade passages are considered as a series of staggered flat plates, the flow is assumed to separate at the TE, forming a wake of zero-velocity and a shear-free isentropic jet, the jet flow exits the blade passage at a zero-deviation angle, compressibility effects are neglected and

finally, flow mixing is assumed to occur downstream the blade LE, introducing losses and changes in the flow direction.

Blockage

The aerodynamic blockage due to reverse flow separation in the passage is assumed to be uniformly distributed along the passage and is given by:

$$b = 1 + \left| \frac{\sin(\beta_2 - \gamma)}{\cos\beta_2} \right| \quad (2.26)$$

Losses

The loss due to mixing-out of the wake and the jet flow at reverse flow separation is computed assuming either constant area (eqn.(2.27)) or constant pressure mixing (eqn.(2.28)) as follows:

$$\bar{\omega}_{mix}^{area} = \frac{\Delta P_0}{1/2\rho V_2^2} = (b^2 - 1)\cos^2\beta_2 \quad (2.27)$$

$$\bar{\omega}_{mix}^{pressure} = \frac{\Delta P_0}{1/2\rho V_2^2} = (b - 1)^2\cos^2\beta_2 \quad (2.28)$$

Deviation

Flow deviation due to mixing-out of the flow is inherently captured by the modified equations of Longley, since the momentum and the kinetic energy of the flow are intrinsically coupled with the blockage parameter. The blockage parameter changes the axial and circumferential momentum as b is generated or dissipated, thereby naturally introducing changes in the flow direction due to mixing-out of the flow. In Longley's code, the above formulations are not implemented since all these phenomena are naturally captured by the inclusion of the blockage parameter in the averaged governing equations and the addition of the blockage-transport equation. At reverse flow conditions, losses are introduced via the turning force, which is projected perpendicular to the blade and not the flow, generating entropy. Additional entropy generation is also introduced due to mixing-out of the flow which results in dissipation of the blockage parameter.

Righi's model

Righi [27, 36, 92] adopted the modelling aspects of Longley's model and presented a model for mixed and reverse flow regimes. His approach is based on the Euler equations and thus does not inherently encompass blockage-mixing effects. These are implicitly accounted for, by implementing different modelling strategies depending on the flow conditions in the blade passage. A flow regime is classified as *mixed regime*, whenever the flow simultaneously enters or exits the blade. A flow regime is classified as *reverse regime*, if the flow enters the blade passage from the TE and exits from the LE. Depending on the regime, different methods are implemented to impose deviation and losses.

Blockage

The aerodynamic blockage due to reverse flow separation in the passage is computed according to Longley's model (eqn.(2.26)). The estimated aerodynamic blockage is assumed to mix-out using a mixing rate similar to Longley's but instead of half a pitch a lengthscale of two pitches is proposed [92]. The predicted aerodynamic blockage is strictly used as an input for the deviation model and does not modify the effective area or volumes of the cells (it is not introducing blockage).

Losses

During reverse or mixed flow operation, losses due to flow separation and mixing-out of the wake and the jet flow, are introduced purely by the turning force. The turning force is projected perpendicular to the blade, introducing losses in the region where the flow angle and the blade passage are not aligned. No other losses are introduced.

Deviation

Flow deviation due to mixing phenomena is estimated by considering the change in the circumferential momentum precipitated by the mixing of the wake and the jet once the flow exits the blade. The deviation is imposed inside the blade passage and is estimated by eqn.(2.29):

$$\tan\beta_1 = \tan\kappa_j \cdot b \quad (2.29)$$

Powers et al. model

The model of Powers et al. [63, 104] is a reduced-order analytical model and does not fall in the through-flow category. However, it entails blockage and loss concepts which could potentially be transferred to through-flow codes, thus they are briefly presented here. At reverse flow conditions, the model accounts for skin friction losses and aerodynamic blockage.

Blockage

Powers et al. [63] proposed an aerodynamic blockage equation for impellers at reverse flow conditions and presented later a revised version of their model. The blockage model is a function of the flow and blade angle at the TE, as well as of a tuning parameter, $\hat{a}$ which represents the intensity of the stall. For a small impeller, the stall parameter was experimentally determined to be 1.7 [63]. The blockage is given by eqn.(2.30) and is expressed in a way so that $b \cdot Area = \text{effective area}$. The blockage parameter expresses the ratio of the effective over the total area, thus $b \leq 1$.

$$b = \frac{1}{\frac{1}{2} \left(\frac{\tan \kappa_{TE}}{\tan \beta_{TE}} + \frac{\tan \beta_{TE}}{\tan \kappa_{TE}} \right) + \hat{a} \left(1 + \frac{1}{2} \frac{\tan \beta_{TE}}{\tan \kappa_{TE}} \right) \left(\frac{\tan \kappa_{TE}}{\tan \beta_{TE}} - 1 \right)^2} \quad (2.30)$$

Losses

At reverse flow conditions, skin friction loss is assumed to be the only source of loss and expresses the wall shear stresses exerted on the fluid due to the no-slip condition. The flow in both the impeller and diffuser is assumed to be strictly radial and the viscous force due to skin friction in impeller and diffuser passages is given by eqn(2.31) and eqn.(2.31) respectively. In impeller passages, the forces are acting strictly in the radial direction:

$$F_{loss} = -\frac{1}{2} f \rho u_r^2 \underbrace{2 \left(\frac{N_b}{2\pi r + \frac{1}{h}} \right)}_{surface} \quad (2.31)$$

In diffuser passages, the forces are acting in both the radial and circumferential direction:

$$F_{loss}^r = -\frac{1}{2}f\rho u_r^2 \underbrace{\frac{2}{h}}_{surface} \quad (2.32)$$

$$F_{loss}^\theta = -\frac{1}{2}f\rho u_r U_\theta \underbrace{\frac{2}{h}}_{surface} \quad (2.33)$$

,where h the blade channel height, N_b the number of impeller blades and f the friction coefficient. The friction coefficient was obtained by curve fitting experimental data and validated against different compressor geometries [63]. It is approximated by eqn.(2.34):

$$f = 0.14 + 5 \cdot 10^{(-6)}\Omega \quad (2.34)$$

Deviation

No deviation is applied.

2.4.3 Blockage modelling methods

Blade blockage, is an essential modelling aspect in through-flow codes which has to be modelled explicitly, due to the absence of distinct blade geometries in the computational domain. Aerodynamic blockage modelling is essential at forward flow conditions primarily in impeller and vaned diffuser blades and is even more important during reverse flow operation due to the separated flow. Aerodynamic blockage is an inherently viscous phenomenon and cannot be naturally reproduced by inviscid solvers. The implications of aerodynamic and metal blockage is the reduction in the effective area, substantially affecting the mass-flow capacity of the compressor (choking) and the overall flow-field. To account for blockage effects, the governing equations are modified, by introducing a blockage parameter, b , which when multiplied by the area or volume of a cell, expresses the effective cell face area or volume. To ensure force equilibrium, blockage gradient source terms are introduced ($P\nabla b$) to account for the pressure forces acting on the blocked areas (no mass flow passes through blocked areas). Implementations of blockage modelling have been performed in the governing equations in the absolute FoR, for instance [38, 49, 85, 94, 96, 103] and a derivation of the modified equations is presented by [38]. The equations inside rotor passages, assuming zero circumferential gradients

(axisymmetric flow-field inside rotors), including blockage and a term which accounts for the rotating of the variables (only in rotor blades) is given by eqn.(2.35). The numerical approximation of the additional blockage gradient terms is not trivial and can result in severe momentum imbalance. Errors in the estimation of the blockage terms near the LE and the TE using finite-difference methods is reported by Kiwada [38]. The implementation of the governing equations with metal and aerodynamic blockage in the relative FoR is not reported in the literature.

$$\begin{aligned}
 \left(\frac{\partial}{\partial t} + \Omega \frac{\partial}{\partial \theta} \right) \begin{bmatrix} b\rho \\ b\rho u_x \\ b\rho u_\theta \\ b\rho u_r \\ b\rho E_0 b \end{bmatrix} + \frac{\partial}{\partial x} \begin{bmatrix} b\rho u_x \\ b\rho u_x^2 + bP \\ b\rho u_\theta u_x \\ b\rho u_r u_x \\ b\rho u_x H_0 \end{bmatrix} + \frac{1}{r} \frac{\partial}{\partial r} \begin{bmatrix} b\rho r u_r \\ b\rho r u_x u_r \\ b\rho r u_\theta u_r \\ b\rho r u_r^2 + Pr \\ b\rho r u_r H_0 \end{bmatrix} \\
 + \begin{bmatrix} 0 \\ 0 \\ \frac{b\rho u_\theta u_r}{r} \\ - \left(\frac{b\rho u_\theta^2}{r} + \frac{bP}{r} \right) \\ b\rho u_\theta H_0 \end{bmatrix} = \begin{bmatrix} 0 \\ bF_x + P \frac{\partial P}{\partial x} \\ bF_\theta \\ bF_r + P \frac{\partial P}{\partial r} \\ b\Omega r F_\theta \end{bmatrix} \quad (2.35)
 \end{aligned}$$

2.5 Research gap

It is evident that over the last decades, body-force modelling approaches have been developed and continuously evolved, ultimately aiming to provide a numerically efficient and reliable alternative approach to traditional CFD turbomachinery methods. These improvements contributed towards ameliorating of their disadvantages, enabling their implementation even in highly complex simulations involving rotating stall and surge events in multi-stage compressors. The majority of the numerical studies in the existing literature are predominantly geared towards axial-flow compressors while in most of those studies, the validity and capability of the models is not extending beyond forward flow regime and stall inception.

Only two, validated through-flow codes capable of pre and post-stall aerodynamic performance modelling are reported in the public domain; Longley's [91] and Righi's [27]. Regarding centrifugal compressor through-flow modelling, the knowledge gap is dramatically exacerbated; only two body force-based modelling attempts for

centrifugal compressors are reported; Benneke's [49] and Kottapalli's [85]. Both are subject to significant limitations, even at forward flow conditions and low rotational speeds and are dependent on extensive CFD input. The first, requires the generation of an extensive database from CFD simulations which is unique to the compressor under consideration. The limitations of the body force database definition approach, the associated interpolation/extrapolation errors and the limitations of the method at transonic regimes are the most essential deficiencies of Benneke's approach. The main limitations of Kottapalli's approach are the high dependency of the body force models on CFD input and the numerical stability issues encountered in 3D simulations which only allowed the acquisition of converged solutions in a 2D, radial impeller. Both methods exhibit large errors near the choking region, while large errors and unrealistic behaviour at near-stall conditions are reported by Benneke. Neither of those methods is able to simulate the post-stall response of centrifugal compressors. The independence of the body force model on external input is a highly desirable feature (explicit model), as well as its capability of accurately imposing the prescribed flow turning and losses transiently, at the presence of high gradients. The body force model of Kottapalli was tested by the author and was deemed inappropriate for such conditions, while the viscous force model of Longley (eqn.(2.12)) was found inadequate for blades with substantial change in the mean streamline curvature. A physically accurate method for distributing losses and flow deviation inside blade passages was not found in the public domain.

Reverse flow modelling methods published in the literature are primarily restricted to axial-flow compressors. The only model applicable to centrifugal compressor configurations, is the one of Powers et al. [63, 104], however it is based on assumptions which are limiting its applicability; mixing effects and the corresponding change in flow angle are neglected, while skin friction is considered to be the only source of dissipation during reverse flow, which is of second order compared to losses due to separation and mixing. The proposed aerodynamic blockage and loss models are functions of coefficients which in the studies, were determined via experimental data. The models of Longley [91] and Righi [27, 36, 92] are derived on the basis of purely axial-flow while considering the blades as staggered flat plates, thereby omitting blade curvature and changes in the inlet and outlet areas. Aerodynamic blockage effects in a steady-state and transient manner are not modelled in any of the aforementioned methodologies, while the governing equations are solved in the absolute FoR. Aerodynamic blockage is expected to constitute an essential modelling aspect at both forward and reverse flow conditions, providing more accurate mass

flow capacity predictions thereby addressing the limitations of [49, 85]. Despite the additional complexity, the transformation of the equations in the relative FoR is expected to yield substantial numerical stability benefits in domains with large rotating domains [105], e.g. impellers. This is a characteristic of particular importance, primarily when simulating full-lengthscale compressor instabilities.

The current project aims, to the extent possible, to propose a novel through-flow modelling approach to address the limitations of the existing methodologies, with the ambition of being sufficiently general, to enable its universal applicability to all types of compressors. Due to the evident knowledge gap in centrifugal compressor through-flow modelling methods and the prior successful implementation of Righi's [27] ideas on axial-flow compressors, the main focus of the PhD was shifted towards the development of the numerical background and methods for centrifugal compressor pre and post-stall modelling.

2.6 Chapter summary

In this chapter, the stall inception mechanisms, full-scale instabilities and different approaches for the modelling of those phenomena are presented. Modal waves and spikes are the two stall precursors underpinning stall inception in both axial and centrifugal compressors, while the latter have only recently been observed experimentally in centrifugal compressors. Modal waves are long-lengthscale instabilities occupying the entire annulus, manifesting several revolutions before the stall event. Spikes emerge suddenly, usually in a single blade passage and subsequently propagate circumferentially, precipitating stall shortly after their emergence. In both axial and centrifugal compressors, the spanwise non-uniformity of the flow seems to be triggering spike inception. In centrifugal compressors, modal waves are inherently related to impeller-vaned diffuser interaction. Rotating stall is a phenomenon whereby stall cells, rotating at a lower speed than the rotor, form around the annulus causing an abrupt loss in pressure rise and efficiency. In centrifugal compressors the pressure characteristic remains continuous and the pressure ratio only mildly reduces. Rotating stall in impellers, vaneless and vaned diffusers is reported. Surge is an axisymmetric phenomenon and the severity of the phenomenon is dictated by rotational speed, plenum of the volume, area and length of the compressor, expressed by the B-parameter. Surge occurs when the volume of the plenum exceeds a critical one. Through-flow methods have been used for decades mainly for forward flow performance predictions. One-dimensional methods are very efficient and rely entirely

on correlations, but do not provide any insight into the flow-field, while are incapable of post-stall predictions. SLC methods rely on empirical models and can resolve the flow-field in both the meridional and blade to blade view, but robustness, incapability of capturing shock waves and dealing with reverse flow are their main deficiencies. 2D, through-flow codes are able of simulating transient forward and reverse flow but cannot capture rotating stall. Only two, out of the four, 2D, through-flow codes found in the literature have demonstrated post-stall capabilities [27, 84], with the first adopting a parallel compressor approach, thus enabling rotating stall simulations and the latter being the axisymmetric version of the full 3D code. No through-flow method applicable to centrifugal compressor is reported. Several body force-based methods have successfully been implemented for axial fans and compressor simulations at forward flow conditions. Gong [86] was the first who successfully captured stall inception and rotating stall phenomena in axial compressors using body forces. While several successful attempts are reported in the literature, only the codes of Longley [91] and Righi [27] have been validated against experimental rotating stall and surge data. Regarding centrifugal compressors, only two, partially successful attempts of (Benneke [49] and Kottapalli [85]) are reported in the public domain but are subject to significant limitations. The first relied entirely on CFD simulations for the extraction of the body forces, while the latter required extensive CFD inputs for the body force models. Very few converged points, on a low-speed characteristic were obtained with the first approach, while the deficiencies of the method were more pronounced in the choking and near-stall region as well as at higher rotational speeds, where no converged simulations could be obtained. The second approach predicted operating points of a 2D, radial impeller, while exhibiting similar inaccuracies at near-choke conditions. No converged solutions were obtained when a 3D centrifugal compressor geometry was modelled with the latter approach. The first, was able of capturing stall inception due to modal waves and spikes but no post-stall response could be simulated with either of the two methods. Body force modelling methods can be classed in 4 main categories: IBMSG, explicit, semi-explicit and CFD-extracted database methods methods. The first, introduces the forces by essentially imposing wall boundary conditions in a regular grid, in the cells of which, information about the local camberline is stored. Explicit models are analytical approaches which do not rely on external input, while semi-explicit ones are analytical models which require external input, typically extracted from CFD simulations. The last approach relies entirely on CFD simulations, whereby all forces are extracted from multiple CFD simulations which are subsequently used to generate a body force-database.

Regarding reverse flow modelling, three models are reported in the literature: Longley's [91], Righi's [27] and the one of Powers et al [33, 63]. The first, accounts for short-lengthscale phenomena by introducing a blockage parameter and a sixth equation, which approximates entropy generation, creation and dissipation of the blockage parameter. It therefore intrinsically reproduces flow deviation, mixing losses and blockage. Simplified models for aerodynamic blockage and mixing losses are also proposed. The second, uses an alternative approach based on Longley's ideas, estimating the change in flow angle via an analytical model, introducing losses entirely via the turning force and omitting reduction in the effective area due to aerodynamic blockage. The latter proposes a blockage parameter and skin friction losses which are functions of tuning parameters. Blockage effects and rotation of the blade passage are modelled by introducing conceptual modifications in the governing equations.

Chapter 3

Methodology

This chapter provides an in-depth discussion in the theory and the different modelling layers underpinning the present work. Initially, the mesh generation strategy is presented along with the considerations regarding its relevance to numerical stability and body force-based applications. The governing equations in both the relative and absolute FoR are presented and the conceptual modifications to account for mixing effects, blockage and rotating domains are analysed. Subsequently, the implemented flux correction method and a new approach for the definition of the additional blockage gradients are discussed. The embedded numerical scheme (Godunov) is coupled with 3 Riemann solvers and their characteristics are analysed. The full blade curvature and blockage distribution is performed on distinct camber, pressure and suction surface grids. A new body force model, comprising a base turning force, correction force and viscous force is developed accounting for the full curvature of the blade and is applicable to all types of compressor blades. The concept of CFD-derived distribution coefficients is introduced, ensuring accurate distribution of losses and deviation in the blade passage. These are coupled with the turning and viscous force models. The assumptions, limitations and theory behind the new blockage-mixing model are analysed. The model provides estimations for aerodynamic blockage, mixing losses and changes in the flow angle due to mixing effects, during reverse flow operation. Subsequently, the underlying modelling strategies for different flow regimes are discussed. The chapter is concluded with a brief description of the 1D plenum-throttle model which allows dynamic post-stall response of the compression system.

Contents

3.1	Computational domain	72
3.2	Solver	77
3.3	Blade metal blockage and curvature definition	106
3.4	Body force model	112
3.5	Blockage-mixing model	129
3.6	Empirical correlations	137
3.7	Forward, mixed and reverse flow modelling	157
3.8	Plenum and throttle model	163
3.9	Chapter summary	166

3.1 Computational domain

The computational domain is structured within the geometric boundaries defined by the hub and shroud aero lines of the compressor duct. Nodes are distributed using an algorithm which automatically defines the position of the vertices enclosing the finite volume cells along the compressor flow path. There are three principal considerations underlying the construction of the grid; numerical stability, the generation of grid topologies which prevent the formation of non-discontinuous body force-fields inside blade passages and the generation of computationally efficient, high-quality, structured grids which prevent the manifestation of non-physical, mesh-related phenomena in the flow-field. Other considerations such as local grid orthogonality, were shown to have a negligible benefit in terms of accuracy and stability for the body-force applications under consideration, subject to the term that cell faces are generally well aligned with the flow-path direction. Grid orthogonality therefore does not constitute an essential but rather an advisable structural aspect of the proposed grid generation approach. Since flow turning and dissipative phenomena are introduced via the body force model, the generation of distinct complex blade grids is completely bypassed, drastically reducing the associated numerical cost e.g. compared to conventional turbomachinery CFD modelling approaches employing turbulence models.

The flow path boundaries are defined by the hub and the shroud axial and radial coordinates (x, r) on the meridional plane. Approximations of the hub and the shroud lines are generated via piecewise cubic interpolation schemes based on the original compressor geometry data. Cubic splines and piecewise cubic Hermite interpolating polynomials (PCHIP) were considered an appropriate interpolation method, capable of generating smooth aero lines while providing a reasonable degree of local curvature without being susceptible to the unpredictable behaviour of high degree polynomial interpolating functions, Bezier curves and other interpolating functions. Their mathematical definition in a piecewise manner also ensures that local interpolating curves always pass through the given set of the duct's local meridional coordinates. Along the shroud and the hub lines, vertices are distributed using a combination of equidistant and variable length node distribution algorithms. Flow regions subject to body-force fields require a larger number of elements in the streamwise direction to ensure numerical stability by gradually turning the flow. Domains outside body-force fields are less prone to numerical instabilities, thus lower resolution grids are preferred in ducted regions to maintain computational cost requirements to a minimum level.

The flow path is either manually divided into subdomains or automatically, following the *adaptive* mesh approach, with the latter not excluding the addition of user-specified domains (*hybrid* mesh). The local mesh resolution of each subdomain is individually specified by varying the number of the cells in the streamwise direction; the number of elements in the spanwise direction is kept constant along the duct. Each of the cells is enclosed by 4 faces (lines), forming arbitrary quadrilaterals along the meridional plane. The mesh generation process is initiated by defining the hub and shroud vertices of each individual subdomain. The nodes are distributed via an equidistant algorithm; the length of the hub and shroud lines enclosing each of the subdomains is computed and subsequently divided by the user-specified number of elements in the streamwise direction corresponding to each of the individual subdomains. In the vicinity of interfaces between adjacent subdomains, a variable node distribution algorithm is activated (ICEM bunching exponential law 1 [106]) to gradually reduce or increase the size of the upstream subdomain elements and match the adjacent domain's cell size. The cell size-matching criteria of adjacent domains is restricted to the streamwise grid lines and is not extended to the spanwise ones. To fully define the computational domain, hub and shroud nodes are linearly connected, forming the grid lines in the spanwise direction. These are divided into equidistant segments forming arbitrary quadrilaterals along the flow path.

3.1.1 Multizone manually-defined grids

The generation of the complete flow path computational domain is performed by synthesizing and merging subdomain grids, the boundaries of which, are defined either manually, automatically or in a hybrid manner. The computational grid can be defined entirely in a manual way by defining the number and the set of coordinates enclosing individual, user-defined subdomains. Each subdomain is defined by two pairs of (x, r) coordinates along the hub and shroud lines coordinates respectively. The number of each of the subdomain's streamwise elements as well the universal/global number of spanwise elements are user-specified mesh parameters. The default node distribution algorithm is an equidistant one, ensuring equally spaced line segments in all directions. Transition regions with variable node distribution in the streamwise direction are optionally introduced, with their length constituting an additional user-specified parameter, defined as fractions of the subdomain enclosing them.

3.1.2 Multizone adaptive grids

Grid generation via the multi-zone, multi-domain approach, enables local mesh refinements inside the more numerically unstable, bladed regions, while also providing the required flexibility to achieve one of the most essential grid features; the positioning of the grid lines in a way which ensures the formation of uniform body-force fields inside bladed regions. This is achieved by aligning the geometric boundaries of the subdomains with the projection of each of the blade's LE and TE on the meridional plane. This results in the formation of distinct subdomains corresponding to strictly ducted or bladed regions and consequently regions with and without body-force fields. Blade boundaries are defined by the meridional coordinates of the LE and TE at the hub and the shroud. Since the grid lines are aligned with the blade boundaries, the grids generated following this approach are referred to here as *adaptive*. Figure(3.1a) shows the subdomains defined automatically during the adaptive mesh generation process of a centrifugal compressor with vaneless diffuser. Figure(3.1b) depicts a hybrid mesh, comprising an additional, user-defined region.

This approach inherently entails the assumption that the meridional projection of the LE and TE of the modelled blades is a linear function, which is a reasonably valid assumption for a wide range of axial compressor, impeller and vaned diffuser blades, but less accurate for modern large fan blades. This could be resolved in the future by accounting for the actual meridional blade curvature at the LE and the TE using the full blade spanwise coordinate data along the LE and the TE. Transition

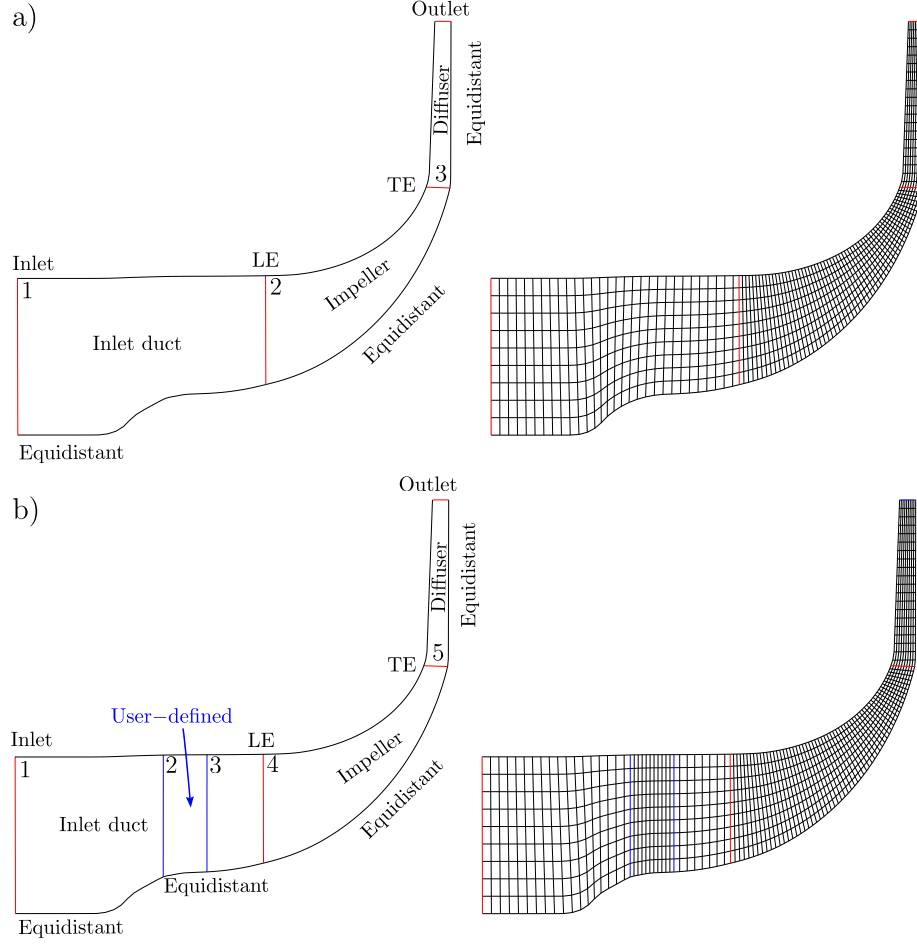


Figure 3.1: a) Adaptive and b) hybrid mesh approach of SRV2-0 centrifugal compressor [107] with inlet bell mouth and vaneless diffuser generated by synthesizing ducted, bladed and user-defined subdomains.

regions can automatically be generated in-between adjacent subdomains but their inclusion inside bladed regions was deemed redundant and potentially detrimental from a numerical stability point of view. In bladed domains, the equidistant algorithm is used exclusively.

Hybrid mesh definition essentially merges manual and adaptive mesh generation approaches incorporating the numerical and modelling advantages of adaptive grids mentioned in section (3.1.2) and the flexibility of user-defined subdomains. The latter enables the definition of lower or higher resolution subdomains providing a means for local grid refinements. Unlike bladed domains, wherein the formation of transition regions is restricted, user-defined domains (similar to ducted ones) may incorporate a combination of equidistant grids with transition regions.

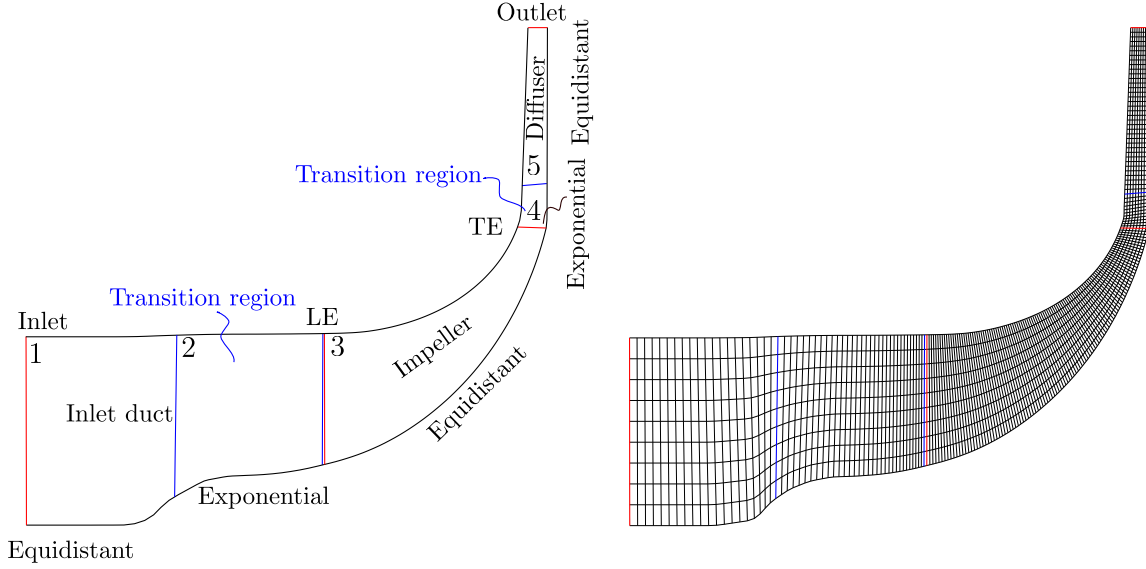


Figure 3.2: Adaptive mesh approach of SRV2-0 centrifugal compressor [107] with the addition of variable node distribution transition regions.

3.1.3 Blade surface grid generation

The evaluation of the blade's spatial curvature and blockage distribution in three dimensions forms the basis of the turning force model which will be discussed in the next section. This is accomplished by first generating blade surface grids of the pressure, suction and camber side. On the nodes of the surface grids, the local blade metal blockage and blade angles describing the curvature of the blades are numerically evaluated.

Pressure and suction 3D coordinates at the hub and the tip is the minimum input requirement to enable the generation of the camber surface mesh coordinates (eqn.(3.77)). An approximation of the blade's aforementioned surface coordinates is subsequently acquired via interpolation methods. Coordinates at additional spanwise positions result in a more precise approximation of the blade surfaces due to smaller interpolation errors, ensuring more accurate curvature estimation. The nodes are generated in sweeps, using piecewise interpolation in three dimensions. Once the user-specified number of stream nodes has been generated (using the given coordinates), a spanwise sweep is performed to obtain the final surface grid. PCHIP is the default interpolation scheme used in the code due to its robustness and ability of generating smoothly curved line segments. The nodes are generated following the equidistant discretisation approach in both the streamwise and spanwise direction. The number of elements in both directions is adjusted separately to ensure sufficient number of elements in high and low aspect ratio blades.

3.2 Solver

The flow-field solution is obtained transiently by solving the inviscid, unsteady compressible Euler equations in either the absolute or relative frame of reference (FoR). The flow-field is considered axisymmetric, therefore no circumferential gradients are present in the governing equations ($\partial/\partial\theta = 0$). The solver is capable of solving the governing equations in both the absolute and the relative (rotating) FoR using a first-order accurate in space and time finite volume scheme. Time-dependent terms are discretised using explicit forward time-stepping, while Godunov's scheme coupled with three Riemann solvers is used to obtain the inter-volume fluxes by solving the local Riemann problem at the faces of the finite volumes. The solver incorporates three Riemann solvers: the HLL, HLLC and Roe's [108]. These are modified accordingly depending on the FoR in which the governing equations are solved. Since no distinct meshed blade topologies are present along the the flow path, the turning of the flow along with the corresponding energy transfer is implicitly introduced via turning force source terms. Viscous dissipation is a substantial modelling aspect, inherently associated with turbomachinery aerodynamic performance. Since shear stresses are not explicitly modelled in the governing equations, viscous-induced losses are imposed via a viscous force model. The reduction in the effective flow-path area due to the presence of blades and aerodynamic-related phenomena is modelled by modifying the governing equations and introducing a blockage parameter, b , which is a function of both the local blade metal and aerodynamic blockage. This constitutes an essential modelling aspect in centrifugal compressors since impeller and vaned diffuser blades introduce substantial metal blockage, while high aerodynamic blockage is encountered even at forward flow conditions. During reverse flow operation, the highly separated flow in the blade passage further exacerbates this phenomenon, significantly affecting the mass flow capacity of the compressor.

3.2.1 Governing equations in the absolute FoR

The unsteady, inviscid, compressible Euler equations can be expressed using absolute velocity components and an additional term, which accounts for the rotation of the flow inside bladed domains. This source term, expressed by a single term, $\rho(\boldsymbol{\Omega} \times \mathbf{u})$ designates the Coriolis and centripetal acceleration contributions [105]. Flow turning and entropy generation are introduced via body force source terms. The energy equation is coupled with a power exchange source term which indicates the energy input into the flow due to the change in angular momentum inside rotors.

The equations can be modified to account for the reduction in effective cell face areas and volumes due to the presence of blade and aerodynamic blockage. The derivation of the equations can be found in [38]. The expanded form of the equations along with the essential conceptual modifications with respect to the simulated component, i.e. ducts, stators and rotors are described in detail by Benneke [49]. Artificial viscosity terms are introduced in the axial and radial momentum equations to provide an artificially-induced mixing and dampening mechanism, since the absence of shear stresses inside inviscid flows may result in unrealistic flow-field phenomena. Air is considered a calorically perfect gas, therefore pressure and temperature specific heat capacities are considered constant and the air conforms with the ideal-gas law. The specific heat ratio, γ is fixed to 1.4 and the ideal gas constant for air, R , is fixed to 287 J/(kgK) . The system is considered adiabatic. The complete solution of the flow-field requires the evaluation of 6 primitive variables; $u_x, u_r, u_\theta, P, \rho$ and T . The flow-field solution is not directly feasible since conservation of continuity, momentum and energy constitute a set of five equations. The system of equations is closed by introducing the ideal gas law equation. The absolute velocity formulation [105] of the unsteady, compressible Euler equations in conservation form is:

Conservation of mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \mathbf{w} = 0 \quad (3.1)$$

Conservation of momentum:

$$\frac{\partial}{\partial t} \rho \mathbf{w} + \nabla \cdot (\rho \mathbf{w} \mathbf{u}) + \rho (\boldsymbol{\Omega} \times \mathbf{u}) = -\nabla P + \mathbf{f} \quad (3.2)$$

Conservation of energy:

$$\frac{\partial}{\partial t} \rho E_0 + \nabla \cdot (\rho \mathbf{w} H_0 + P (\boldsymbol{\Omega} \times \mathbf{r})) = \dot{W}_{ex} \quad (3.3)$$

Ideal gas law:

$$P = \rho RT \quad (3.4)$$

In the above equations, $\mathbf{w}$ is the relative velocity vector, $\mathbf{u}$ is the absolute velocity vector, ρ the density, P the static pressure, $\mathbf{\Omega}$ the angular velocity vector, $\mathbf{r}$ the radial position vector, E_r the specific relative internal energy, $\mathbf{f}$ the body force vector and $\dot{W}_{ex}$ the power exchange source term. The relative and absolute velocity vectors assuming constant angular velocity and rotation axis parallel to the symmetry axis (x-axis) are related via:

$$\mathbf{w} = \mathbf{u} - \mathbf{\Omega} \times \mathbf{r} \quad (3.5)$$

Which in expanded form are given by:

$$\begin{bmatrix} w_x \\ w_\theta \\ w_r \end{bmatrix} = \begin{bmatrix} u_x \\ u_\theta - \Omega r \\ u_r \end{bmatrix} \quad (3.6)$$

The specific total energy, E_0 and total enthalpy, H_0 are evaluated as follows:

$$E_0 = e + \frac{1}{2}V^2 = c_v T + \frac{1}{2}V^2 = \frac{P}{\rho} \frac{1}{(\gamma - 1)} + \frac{1}{2}V^2 \quad (3.7)$$

$$H_0 = h + \frac{1}{2}V^2 = c_p T + \frac{1}{2}V^2 = \frac{P}{\rho} \frac{\gamma}{(\gamma - 1)} + \frac{1}{2}V^2 \quad (3.8)$$

,where

$$V = \sqrt{u_x^2 + u_\theta^2 + u_r^2} \quad (3.9)$$

The power exchange source term, $\dot{W}_{ex}$ is the product of the blade speed and the circumferential body force component:

$$\dot{W}_{ex} = \Omega r f_\theta \quad (3.10)$$

3.2.2 Governing equations inside stationary and rotating components

The flow is considered axisymmetric, thus circumferential gradients and fluxes are omitted from the analysis ($\partial/\partial\theta = 0$). With respect to the component under consideration i.e. duct, stator or rotor, conceptual modifications are essential to ensure consistency with the modelling assumptions underlying each component e.g. blockage, mixing and body forces. Flow mixing is an essential flow-feature and is modelled via the addition of artificial diffusion terms. These terms are always included in the governing equations regardless of the component under consideration and are discussed in detail in section(3.2.6). The effect of aerodynamic and blade blockage is strictly restricted to bladed domains; ducted domains are always solved using the full annulus areas and volumes of the finite volume cells. Since the entire computational domain is modelled as a 2D axisymmetric duct, distinct meshed blade geometries are not present and consequently do not interfere with the computational domain and thereby the flow-field solution.

The reduction in the effective area due to the finite thickness of the blades and aerodynamic blockage, needs to be implicitly accounted for, via a reduction in the effective areas and volumes of the finite volume cells positioned inside bladed regions. A blockage parameter, b , is introduced in the governing equations, the value of which, ranges between zero and one. The blockage parameter is the ratio of the effective over the entire area/volume; a blockage parameter equal to one designates a cell face area/volume without blockage while a blockage parameter equal to zero, corresponds to a fully blocked one. Body forces are exerted on cells lying inside bladed regions but are also exerted on ducted domain cells at reverse flow conditions to impose additional losses due to mixing-out of the flow.

Ducted components

Inside ducted domains the effective area is considered equal to the geometric one regardless of the flow conditions. Losses are introduced via a source term at reverse flow conditions while no source terms are introduced at forward flow conditions. The artificial viscosity terms and the corresponding nomenclature is discussed in section(3.2.6). The equations inside ducted domains are:

$$\begin{aligned}
\frac{\partial}{\partial t} \begin{bmatrix} \rho \\ \rho u_x \\ \rho u_\theta \\ \rho u_r \\ \rho E_0 \end{bmatrix} + \frac{\partial}{\partial x} \begin{bmatrix} \rho u_x \\ \rho u_x^2 + P \\ \rho u_\theta u_x \\ \rho u_r u_x \\ \rho u_x H_0 \end{bmatrix} + \frac{1}{r} \frac{\partial}{\partial r} \begin{bmatrix} \rho r u_r \\ \rho r u_x u_r \\ \rho r u_\theta u_r \\ \rho r u_r^2 + Pr \\ \rho r u_r H_0 \end{bmatrix} + \begin{bmatrix} 0 \\ 0 \\ \frac{\rho u_\theta u_r}{r} \\ -\left(\frac{\rho u_\theta^2}{r} + \frac{P}{r}\right) \\ \rho u_\theta H_0 \end{bmatrix} = \begin{bmatrix} 0 \\ f_x \\ f_\theta \\ f_r \\ 0 \end{bmatrix} + \begin{bmatrix} 0 \\ f_{\tau,x} \\ 0 \\ f_{\tau,r} \\ \dot{W}_\tau \end{bmatrix}
\end{aligned} \tag{3.11}$$

Stationary bladed components

The presence of non-rotating bladed components e.g. OGV's, IGV's, axial stators, vaned diffuser blades etc. results in a reduction in the effective cell face area and volumes dictated by the local blade and aerodynamic blockage distributions. Flow turning and losses are modelled via the inclusion of body force source terms in the momentum equations. The power input term is set to zero since there is no energy exchange in stator blades. The equations inside stationary bladed domains are:

$$\begin{aligned}
\frac{\partial}{\partial t} \begin{bmatrix} b\rho \\ b\rho u_x \\ b\rho u_\theta \\ b\rho u_r \\ b\rho E_0 b \end{bmatrix} + \frac{\partial}{\partial x} \begin{bmatrix} b\rho u_x \\ b\rho u_x^2 + bP \\ b\rho u_\theta u_x \\ b\rho u_r u_x \\ b\rho u_x H_0 \end{bmatrix} + \frac{1}{r} \frac{\partial}{\partial r} \begin{bmatrix} b\rho r u_r \\ b\rho r u_x u_r \\ b\rho r u_\theta u_r \\ b\rho r u_r^2 + Pr \\ b\rho r u_r H_0 \end{bmatrix} + \begin{bmatrix} 0 \\ 0 \\ \frac{b\rho u_\theta u_r}{r} \\ -\left(\frac{b\rho u_\theta^2}{r} + \frac{bP}{r}\right) \\ b\rho u_\theta H_0 \end{bmatrix} \\
= \begin{bmatrix} 0 \\ bf_x \\ bf_\theta \\ bf_r \\ 0 \end{bmatrix} + \begin{bmatrix} 0 \\ bf_{\tau,x} \\ 0 \\ bf_{\tau,r} \\ b\dot{W}_\tau \end{bmatrix} + \begin{bmatrix} 0 \\ P\frac{\partial b}{\partial x} \\ 0 \\ P\frac{\partial b}{\partial r} \\ 0 \end{bmatrix}
\end{aligned} \tag{3.12}$$

Rotating bladed components

The modelling approach of the flow inside rotor blades adopts the same conceptual modifications applied to stator blades, with the only differentiation being the addition of the power exchange term to account for the addition of energy into the flow. This energy input manifests in the form of total enthalpy increments as the flow passes through a rotor's body-force field. The equations inside rotating bladed components are:

$$\begin{aligned}
 \frac{\partial}{\partial t} \begin{bmatrix} b\rho \\ b\rho u_x \\ b\rho u_\theta \\ b\rho u_r \\ b\rho E_0 b \end{bmatrix} + \frac{\partial}{\partial x} \begin{bmatrix} b\rho u_x \\ b\rho u_x^2 + bP \\ b\rho u_\theta u_x \\ b\rho u_r u_x \\ b\rho u_x H_0 \end{bmatrix} + \frac{1}{r} \frac{\partial}{\partial r} \begin{bmatrix} b\rho r u_r \\ b\rho r u_x u_r \\ b\rho r u_\theta u_r \\ b\rho r u_r^2 + Pr \\ b\rho r u_r H_0 \end{bmatrix} + \begin{bmatrix} 0 \\ 0 \\ \frac{b\rho u_\theta u_r}{r} \\ -\left(\frac{b\rho u_\theta^2}{r} + \frac{bP}{r}\right) \\ b\rho u_\theta H_0 \end{bmatrix} \\
 = \begin{bmatrix} 0 \\ bf_x \\ bf_\theta \\ bf_r \\ b\dot{W}_{ex} \end{bmatrix} + \begin{bmatrix} 0 \\ bf_{\tau,x} \\ 0 \\ bf_{\tau,r} \\ b\dot{W}_\tau \end{bmatrix} + \begin{bmatrix} 0 \\ P\frac{\partial b}{\partial x} \\ 0 \\ P\frac{\partial b}{\partial r} \\ 0 \end{bmatrix} \quad (3.13)
 \end{aligned}$$

3.2.3 Governing equations in the relative FoR

An alternative form of the unsteady, inviscid, compressible Euler equations can be derived by expressing the governing equations in the relative FoR. Since the governing equations are derived in the rotating FoR, additional body force source terms are introduced to conserve momentum in the circumferential and radial direction. These source terms correspond to Coriolis and centrifugal forces respectively and are fictitious; they purely arise due to the formulation of the governing equations in a non-inertial, rotating FoR. The energy equation expresses the conservation of rothalpy along a relative streamline, therefore the power exchange source term is omitted in the energy equation.

To ensure mass, momentum and energy conservation, flux correction is applied to interfaces between adjacent domains with different rotational speeds. Flux correction is indispensable when the flow-field is solved in the relative FoR, since the rotational

velocity component and its corresponding kinetic energy are not perceivable in the rotating FoR. As discussed earlier, air is considered an ideal gas and the system of equations is closed by introducing the ideal gas law (eq.(3.17))

Relative velocity formulation

The unsteady, compressible Euler equations expressed in conservation form and cylindrical coordinate system can be written in vectorial form as follows [105]:

Conservation of mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{w}) = 0 \quad (3.14)$$

Conservation of momentum:

$$\frac{\partial}{\partial t} (\rho \mathbf{w}) + \nabla \cdot (\rho \mathbf{w} \mathbf{w} + P) + \rho (2\boldsymbol{\Omega} \times \mathbf{w} + \boldsymbol{\Omega} \times (\boldsymbol{\Omega} \times \mathbf{r})) = \rho \mathbf{f} \quad (3.15)$$

Conservation of energy:

$$\frac{\partial}{\partial t} (\rho E_r) + \nabla \cdot (\rho \mathbf{w} I) = 0 \quad (3.16)$$

Ideal gas law:

$$P = \rho RT \quad (3.17)$$

In the above equations, $\mathbf{w}$ is the relative velocity vector, ρ the density, P the static pressure, $\boldsymbol{\Omega}$ the angular velocity vector, $\mathbf{r}$ the radial position vector, E_r the specific relative internal energy, I the rothalpy and $\mathbf{f}$ the body force vector. The specific relative internal energy and rothalpy terms can be evaluated by expressing the specific internal energy, e as a function of either the temperature or the pressure as follows:

$$E_{rel} = e + \frac{1}{2} (W^2 - u_{rot}^2) = c_V T + \frac{1}{2} (W^2 - u_{rot}^2) = \frac{P}{\rho} \frac{1}{\gamma - 1} + \frac{1}{2} (W^2 - u_{rot}^2) \quad (3.18)$$

$$I = E_{rel} + \frac{P}{\rho} = c_P T + \frac{1}{2} W^2 - u_{rot}^2 = \frac{P}{\rho} \frac{\gamma}{\gamma - 1} + \frac{1}{2} (W^2 - u_{rot}^2) \quad (3.19)$$

Relative velocity formulation with blockage

Blockage effects are numerically introduced in the governing equations following a similar approach to the one presented earlier. Even though momentum and mass transfer do not take place through blocked sectors, pressure forces are exerted and need to be considered to prevent momentum imbalance. These pressure forces arise in the form of blockage gradient source terms in the axial and radial momentum equations and were also present in the absolute velocity formulation. Metal blockage is independent of the circumferential position thus no additional terms arise in the circumferential momentum equation. The blockage parameter is a function of both the local metal and aerodynamic blockage parameters with the latter being updated conjointly with the corresponding blockage gradient source terms in a transient manner. The derivation of the equations has been performed following a similar approach to that of Kiwada [38] and is included in the appendix(6.4). The equations in vectorial form are given below:

Conservation of mass:

$$\frac{\partial b\rho}{\partial t} + \nabla \cdot (b\rho \mathbf{w}) = 0 \quad (3.20)$$

Conservation of momentum:

$$\frac{\partial}{\partial t} (b\rho \mathbf{w}) + \nabla \cdot (b\rho \mathbf{w} \mathbf{w} + bP) + b\rho (2\boldsymbol{\Omega} \times \mathbf{w} + \boldsymbol{\Omega} \times (\boldsymbol{\Omega} \times \mathbf{r})) = P\nabla b + \rho \mathbf{F} \quad (3.21)$$

Conservation of energy:

$$\frac{\partial}{\partial t} (b\rho E_r) + \nabla \cdot (b\rho \mathbf{w} I) = 0 \quad (3.22)$$

3.2.4 Governing equations in ducted and bladed domains

The governing equations in the stationary and rotating FoR with and without blockage, form the mathematical basis to resolve the flow-field along the discretised flow-path domain. The flow along the entire computational domain is considered axisymmetric, i.e. $(\partial/\partial\theta = 0)$ and adiabatic. Identical conceptual modifications in the governing equations are introduced to ensure consistency with the aforementioned modelling approaches underpinning bladed and ducted domains. When the equations are solved in the rotating FoR, ducted and stationary blade components are solved in the absolute FoR, since the angular velocity is zero ($\Omega = 0$) and both FoR coincide. Rotating components are solved in the relative FoR, while flux corrections are required at interfaces adjoining cells with different rotational velocities. In an adaptive grid, these interfaces coincide with the LE and TE of the blades.

Ducted components

Inside ducted regions, there is no distinction between absolute and relative FoR ($\Omega = 0$) and the equations reduce to the absolute velocity formulation. Metal and aerodynamic blockage is neglected. Depending on the flow regime, losses are introduced via body force source terms. The equations inside ducted domains are:

$$\begin{aligned}
 \frac{\partial}{\partial t} \begin{bmatrix} \rho \\ \rho u_x \\ \rho u_\theta \\ \rho u_r \\ \rho E_0 \end{bmatrix} + \frac{\partial}{\partial x} \begin{bmatrix} \rho u_x \\ \rho u_x^2 + P \\ \rho u_\theta u_x \\ \rho u_r u_x \\ \rho u_x H_0 \end{bmatrix} + \frac{1}{r} \frac{\partial}{\partial r} \begin{bmatrix} \rho r u_r \\ \rho r u_x u_r \\ \rho r u_\theta u_r \\ \rho r u_r^2 + P r \\ \rho r u_r H_0 \end{bmatrix} + \begin{bmatrix} 0 \\ 0 \\ \frac{\rho u_\theta u_r}{r} \\ -\left(\frac{\rho u_\theta^2}{r} + \frac{P}{r}\right) \\ \rho u_\theta H_0 \end{bmatrix} = \begin{bmatrix} 0 \\ f_x \\ f_\theta \\ f_r \\ 0 \end{bmatrix} + \begin{bmatrix} 0 \\ f_{\tau,x} \\ 0 \\ f_{\tau,r} \\ \dot{W}_\tau \end{bmatrix}
 \end{aligned}
 \tag{3.23}$$

Stationary bladed components

Inside stationary, bladed domains e.g. axial stators, IGVs, vaned diffuser blades etc., the reduction in the effective area is modelled by solving the modified version of the governing equations in the absolute FoR (eqn.(3.24)), since $\Omega = 0$ and the relative velocity formulation reduces to the absolute one. Entropy generation and flow turning are introduced via body force source terms, while flow mixing is achieved via artificial diffusion source terms. The governing equations in stator domains have been presented in the previous section but are included here for the sake of completeness:

$$\begin{aligned}
 \frac{\partial}{\partial t} \begin{bmatrix} b\rho \\ b\rho u_x \\ b\rho u_\theta \\ b\rho u_r \\ b\rho E_0 b \end{bmatrix} + \frac{\partial}{\partial x} \begin{bmatrix} b\rho u_x \\ b\rho u_x^2 + bP \\ b\rho u_\theta u_x \\ b\rho u_r u_x \\ b\rho u_x H_0 \end{bmatrix} + \frac{1}{r} \frac{\partial}{\partial r} \begin{bmatrix} b\rho r u_r \\ b\rho r u_x u_r \\ b\rho r u_\theta u_r \\ b\rho r u_r^2 + Pr \\ b\rho r u_r H_0 \end{bmatrix} + \begin{bmatrix} 0 \\ 0 \\ \frac{b\rho u_\theta u_r}{r} \\ -\left(\frac{b\rho u_\theta^2}{r} + \frac{bP}{r}\right) \\ b\rho u_\theta H_0 \end{bmatrix} \\
 = \begin{bmatrix} 0 \\ bf_x \\ bf_\theta \\ bf_r \\ 0 \end{bmatrix} + \begin{bmatrix} 0 \\ bf_{\tau,x} \\ 0 \\ bf_{\tau,r} \\ b\dot{W}_\tau \end{bmatrix} + \begin{bmatrix} 0 \\ P\frac{\partial b}{\partial x} \\ 0 \\ P\frac{\partial b}{\partial r} \\ 0 \end{bmatrix} \quad (3.24)
 \end{aligned}$$

Rotating bladed components

Inside rotating bladed domains, the relative velocity formulation of the Euler equations with blockage is used. Artificial shear stress terms are added, while body force source terms are introduced in the momentum equations. In the rotating FoR, no power exchange between the rotors and the flow takes place. Blockage gradient terms, identical to the ones in the absolute FoR arise due to the inclusion of the blockage parameters in the governing equations. All body forces, including the centripetal and Coriolis terms are multiplied by the blockage parameter. The governing equations inside rotor blades in the rotating FoR are:

$$\begin{aligned}
& \frac{\partial}{\partial t} \begin{bmatrix} b\rho \\ b\rho w_x \\ b\rho w_\theta \\ b\rho w_r \\ b\rho E_{rel} \end{bmatrix} + \frac{\partial}{\partial x} \begin{bmatrix} b\rho w_x \\ b\rho w_x^2 + P \\ b\rho w_\theta w_x \\ b\rho w_r w_x \\ b\rho w_x I \end{bmatrix} + \frac{1}{r} \frac{\partial}{\partial r} \begin{bmatrix} b\rho r w_r \\ b\rho r w_x w_r \\ b\rho r w_\theta w_r \\ b\rho r w_r^2 + Pr \\ b\rho r w_r I \end{bmatrix} + \begin{bmatrix} 0 \\ 0 \\ \frac{b\rho w_\theta w_r}{r} \\ -\left(\frac{b\rho w_\theta^2}{r} + \frac{bP}{r}\right) \\ 0 \end{bmatrix} \\
& + \begin{bmatrix} 0 \\ 0 \\ 2b\rho\Omega w_r \\ -b\rho(2\Omega w_\theta + \Omega^2 r) \\ 0 \end{bmatrix} = \begin{bmatrix} 0 \\ bf_x \\ bf_\theta \\ bf_r \\ 0 \end{bmatrix} + \begin{bmatrix} 0 \\ bf_{\tau,x} \\ 0 \\ bf_{\tau,r} \\ b\dot{W}_\tau \end{bmatrix} + \begin{bmatrix} 0 \\ P\frac{\partial b}{\partial x} \\ 0 \\ P\frac{\partial b}{\partial r} \\ 0 \end{bmatrix} \quad (3.25)
\end{aligned}$$

3.2.5 Flux corrections at interfaces

Solution of the flow-field in the relative FoR requires flux correction at interfaces adjoining cells with different angular velocities to prevent erroneous evaluation of the momentum and energy fluxes. Inside rotor passages, the rotational kinetic components and their corresponding flux contributions are not perceivable in the relative FoR. When the flow exits a rotor, the distinction between relative and absolute FoR is irrelevant and the circumferential and energy fluxes comprise the contribution of these additional components. The aforementioned flux contributions are purely restricted to the energy and circumferential momentum fluxes, subject to the condition, that the axis of rotation coincides with the axis of symmetry (x-axis). This is a valid assumption for most turbomachinery applications. To solve the local Riemann problem at interfaces adjoining domains of different rotational velocities, two fluxes are locally obtained; one in the absolute and one in the relative FoR. The governing equations are subsequently solved using the absolute FoR interface flux if the cell is within the stationary domain and the corresponding relative FoR interface flux, if the cell is positioned within the rotating domain.

The convection term expressed in the absolute and relative velocity formulation respectively is:

$$Flux_{mom,abs} = \nabla \cdot (\rho \mathbf{u} \mathbf{u}) \quad (3.26)$$

$$Flux_{mom,rel} = \nabla \cdot (\rho \mathbf{w} \mathbf{w}) \quad (3.27)$$

The energy flux term expressed in the absolute and relative FoR respectively is:

$$Flux_{E,rel} = \nabla \cdot (\rho \mathbf{w} I) \quad (3.28)$$

$$Flux_{E,abs} = \nabla \cdot (\rho \mathbf{u} H_0) \quad (3.29)$$

Assuming that the rotation axis and the axis of symmetry coincide, the axial and radial velocity components and consequently the corresponding momentum fluxes, are identical regardless of the FoR. This does not apply to the circumferential momentum and energy fluxes since the angular velocity of the rotor is not perceivable in the relative FoR. The circumferential momentum and energy fluxes for axisymmetric flow in the relative FoR can be transformed to their equivalent flux in the absolute FoR as follows:

Circumferential flux correction:

$$\frac{Flux_{\theta,rel}}{Flux_{\theta,abs}} = \frac{\rho w_{\theta} w_{f,n}}{\rho u_{\theta} u_{f,n}} = \frac{u_{\theta} - u_{rot}}{u_{\theta}} \quad (3.30)$$

The circumferential momentum flux transformation between the stationary and rotating FoR is given by:

$$\boxed{Flux_{\theta,rel} = Flux_{\theta,abs} \left(1 - \frac{u_{rot}}{u_{\theta}} \right)} \quad (3.31)$$

Energy flux correction:

$$\frac{Flux_{energy,rel}}{Flux_{energy,abs}} = \frac{\rho w_{f,n} I}{\rho u_{f,n} H_0} = \frac{H_0 - u_{rot} u_{\theta}}{H_0} \quad (3.32)$$

The energy flux transformation between the stationary and rotating FoR is given by:

$$\boxed{Flux_{energy,rel} = Flux_{energy,abs} \left(1 - \frac{u_{rot} u_{\theta}}{H_0} \right)} \quad (3.33)$$

In the above equations $w_{f,n} = u_{f,n}$ are the velocity components normal to the cell faces. These are acting on the meridional plane and are identical in both the

rotating and the absolute FoR. The solution of the Riemann problem at cell faces adjoining stationary and rotating domains is performed by computing 2 fluxes; one in the absolute and one in the relative FoR. To update the flow-field solution, interface fluxes at the absolute FoR are used if the finite volume centroid is positioned inside a stationary domain and relative ones when the finite volume centroid is positioned inside a rotating domain i.e. cells upstream the TE. The corrected fluxes ensure consistency with respect to the frame of reference, since the equations always reduce to the absolute velocity formulation inside stationary domains. Figure (3.3) schematically illustrates the transformation of the fluxes at the interface adjoining a stationary and a rotating domain.

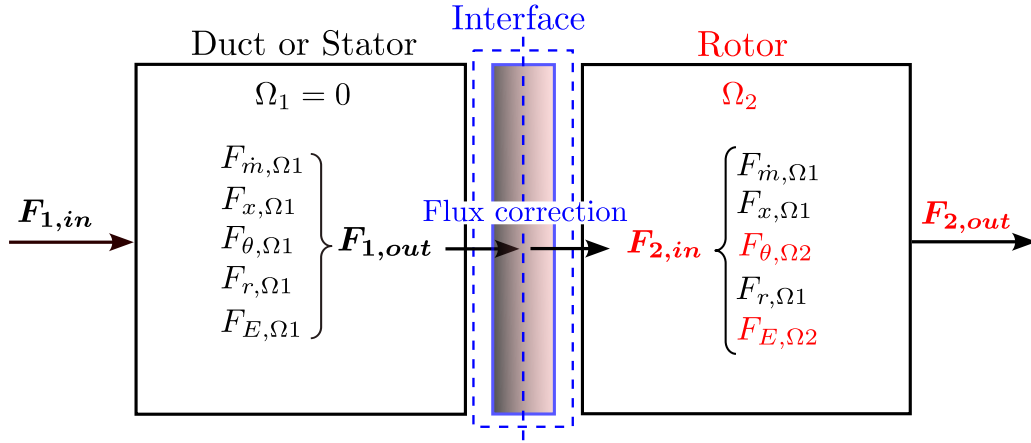


Figure 3.3: Flux correction at the interface adjoining stationary and rotating domains.

3.2.6 Artificial diffusion source terms

A significant modelling aspect which is not explicitly modelled due to the assumption of inviscid flow, is the absence of shear stresses between adjacent fluid layers convecting with different momentum. The implication of this assumption is the inability of momentum and energy transfer across fluid layers and therefore of flow mixing. This is not expected to substantially influence the flow-field and the aerodynamic performance predictions at forward flow conditions since the flow is expected to be rather uniform, without large separation regions and large velocity gradients in the streamwise and spanwise direction. However, mixed and reverse flow regimes are expected to be dominated by large velocity gradients, extensive separation regions and non-uniformity. During transient surge cycle simulations, unrealistically prolonged instabilities were observed due to lack of a dampening, mixing mechanism. An artificially-imposed mixing mechanism is induced by introducing artificial

shear stress terms, expressed as second order derivatives of streamwise and spanwise velocities in the streamwise and spanwise directions. Since the flow is considered axisymmetric, circumferential mixing modelling is redundant. The additional terms are multiplied by an artificial kinematic viscosity coefficient and are introduced in the energy as well as the axial and radial momentum equations, enabling momentum and energy transfer along the meridional plane.

The shear stresses acting in the streamwise and spanwise direction τ_{mn}, τ_{nm} , are considered to be equal to ensure that no moment is exerted on the fluid [109]. A correction is applied to account for the change in the areas of the cell faces. The shear stresses are proportional to the shear strain rate of the fluid and a user-specified kinematic viscosity coefficient, ν^* and are approximated as follows:

$$\tau_{mn,f} = \tau_{nm,f} = \nu^* \left(\frac{\partial u_n}{\partial m} + \frac{\partial u_m}{\partial n} \right) \quad (3.34)$$

The shear stress gradient terms result in second order single and mixed derivative terms with respect to the streamwise and spanwise grid coordinates. Their contributions in the axial and radial direction is approximated by projecting the contributions of those terms along the axial and radial axes:

Axial momentum artificial diffusion term:

$$\rho \nu^* \left[\cos \phi_{f,m} \left(\frac{\partial^2 u_{n,f}}{\partial n_f \partial m_f} + \frac{\partial^2 u_{m,f}}{\partial n_f^2} \right) + \sin \phi_{f,m} \left(\frac{\partial^2 u_{m,f}}{\partial n_f \partial m_f} + \frac{\partial^2 u_{n,f}}{\partial m_f^2} \right) \right] \quad (3.35)$$

Radial momentum artificial diffusion term:

$$\rho \nu^* \left[-\sin \phi_{f,m} \left(\frac{\partial^2 u_{n,f}}{\partial n_f \partial m_f} + \frac{\partial^2 u_{m,f}}{\partial n_f^2} \right) + \cos \phi_{f,m} \left(\frac{\partial^2 u_{m,f}}{\partial n_f \partial m_f} + \frac{\partial^2 u_{n,f}}{\partial m_f^2} \right) \right] \quad (3.36)$$

The streamwise and spanwise velocity terms are expressed with respect to the geometrical, grid-defined streamwise and spanwise coordinate system; the first is parallel, while the latter perpendicular to the face vectors of the cells facing in

the streamwise direction. This facilitates the evaluation of these terms since the velocities along the grid-defined directions are known, thus bypassing the need for approximating gradients in other directions using interpolation schemes. It also allows for a distinction between positive and negative streamwise and spanwise velocities with negative velocities designating flow parallel but opposite to the local cell face vector. The energy flux associated with the artificial shear stress terms is the product of the artificial shear stresses and the corresponding velocity components along the grid coordinate system. Expanding terms using chain rule and setting $\tau_{mn} = \tau_{nm}$ (eqn.3.34), the shear stress energy flux terms are expressed as a function of first and second order derivatives:

Energy transfer artificial diffusion term:

$$\left(\frac{(u_{m,f}\tau_{mn,f})}{\partial n_f} + \frac{(u_{n,f}\tau_{nm,f})}{\partial m_f} \right) = \tau_{mn,f} \left(\frac{\partial u_{m,f}}{\partial n_f} + \frac{\partial u_{n,f}}{\partial m_f} \right) + u_{m,f} \frac{\partial \tau_{mn,f}}{\partial n_f} + u_{n,f} \frac{\partial \tau_{mn,f}}{\partial m_f} \quad (3.37)$$

Substituting the shear stress term, τ_{mn} with eqn.(3.34) the energy term can be written as follows:

$$\rho\nu^* \left[\left(\frac{\partial u_{m,f}}{\partial n_f} + \frac{\partial u_{n,f}}{\partial m_f} \right)^2 + u_{m,f} \left(\frac{\partial^2 u_{n,f}}{\partial n_f \partial m_f} + \frac{\partial^2 u_{m,f}}{\partial n_f^2} \right) + u_{n,f} \left(\frac{\partial^2 u_{m,f}}{\partial n_f \partial m_f} + \frac{\partial^2 u_{n,f}}{\partial m_f^2} \right) \right] \quad (3.38)$$

In the above equations, $\phi_{f,m}$ is the cell face vector angle with respect to the x-axis, m_f and n_f the streamwise and spanwise coordinates with respect to the cell faces, ν^* the artificial kinematic viscosity, τ_{mn} the shear stress acting on the plane normal to m_f and direction parallel to n_f , τ_{nm} the shear stress acting on the plane normal to n_f and direction parallel to m_f , $u_{m,f}$ the velocity component normal to the cell face and $u_{n,f}$ the velocity component parallel to the cell face. The velocity components $u_{m,f}$ and $u_{n,f}$ are given by eqns.(3.39) and (3.40) respectively:

$$u_{m,f} = \cos\phi_{f,m}u_x + \sin\phi_{f,m}u_r \quad (3.39)$$

$$u_{f,n} = -\sin\phi_{f,m}u_x + \cos\phi_{f,m}u_r \quad (3.40)$$

The contribution of the mixed derivative terms was found to be comparatively of lesser importance with respect to the other terms. Preliminary simulations to assess the ability of both formulations to replicate mixing phenomena in flows with high streamwise and spanwise velocity gradients showed that omission of those terms yielded similar results. Further assessment of the artificial viscosity terms and the formulations is required in the future. The reduced version of eqns.(3.35)-(3.38) implemented in the governing equations is obtained by eliminating the mixed derivative terms:

Axial momentum artificial diffusion term:

$$\rho\nu^* \left(\cos\phi_m \frac{\partial^2 V_m}{\partial n^2} - \sin\phi_m \frac{\partial^2 V_n}{\partial m^2} \right) \quad (3.41)$$

Radial momentum artificial diffusion term:

$$\rho\nu^* \left(\sin\phi \frac{\partial^2 V_m}{\partial n^2} + \cos\phi \frac{\partial^2 V_n}{\partial m^2} \right) \quad (3.42)$$

Energy transfer artificial diffusion term:

$$\rho\nu^* \left[\left(\frac{\partial u_{m,f}}{\partial n_f} + \frac{\partial u_{n,f}}{\partial m_f} \right)^2 + u_{m,f} \frac{\partial^2 u_{m,f}}{\partial n_f^2} + u_{n,f} \frac{\partial^2 u_{n,f}}{\partial m_f^2} \right] \quad (3.43)$$

3.2.7 Riemann solvers

The solution of the flow-field is obtained transiently using Godunov's scheme, a first order accurate in space and time, monotone, finite volume scheme. Flow-field properties inside finite volume cells are considered uniform i.e. a piecewise constant distribution is assumed inside discrete finite volume cells. Higher order schemes, e.g.

MUSCL (Monotonic Upwind Scheme for Conservation Laws) approximate flow-field variations inside cells using linear or higher order interpolating functions but are not inherently monotonic, requiring flux limiters to prevent overshooting and numerical instabilities [108]. The scheme evaluates mass, momentum and energy fluxes at each cell face by solving the local Riemann problem at each inter-cell boundary. The local Riemann problem stems from the discontinuous flow-field properties present at each individual cell boundary. The discontinuity naturally manifests whenever the left and the right flow-field states at an inter-cell boundary differ; these are dictated by the upstream and downstream flow-field properties of the adjacent cells.

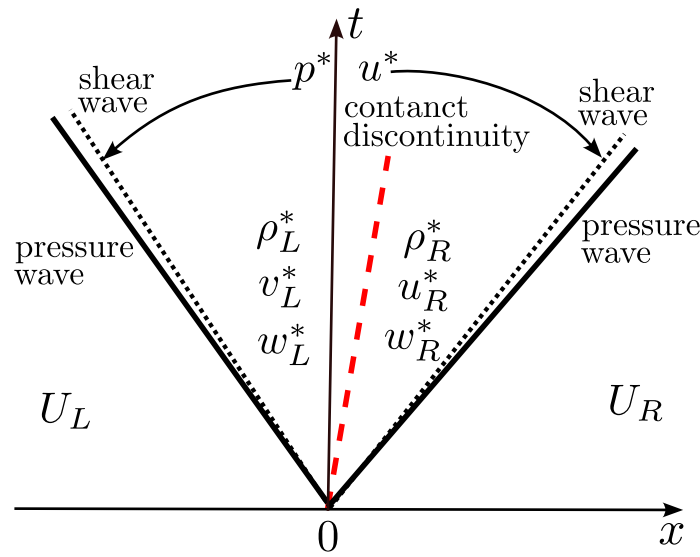


Figure 3.4: Wave structure of the solution of the exact Riemann problem of the x-split, 3D Euler equations (adapted from [108])

The structure of the Riemann problem comprises a combination of different types of pressure waves; these can be non-linear waves i.e. shock waves or expansion fans, contact discontinuities and shear waves. The leftmost and rightmost waves are always associated with non-linear waves i.e. shock waves or expansion fans, while the intermediate waves consist of 2 shear waves and a contact discontinuity. Contrary to approximate Riemann solvers, exact ones account for the presence of all waves, providing more accurate solutions, but the pressure is determined iteratively e.g. using Newton-Raphson method. The stability and numerical efficiency of the solution scheme (e.g. Newton-Raphson) is dependent upon the accuracy of the initial pressure estimation. Inaccurate initial pressure estimations result in slower numerical convergence of the Newton-Raphson solver and might potentially lead to divergence of the scheme. The implications of the aforementioned limitations are detrimental in

terms of both numerical cost and numerical stability.

The accuracy benefits of exact Riemann solvers was assessed and was deemed insufficient to justify the additional complexity, numerical cost and robustness constraints they imply. The solver is coupled with 3 Riemann solvers; the HLL (Harten, Lax, van Leer), HLLC (Harten, Lax, van Leer, Contact) and the linearised solver of Roe (Roe-Pike variant). The evaluation of other improved variants of the HLL Riemann solver, such as the HLLE, HLLM and HLLME [108, 110] was deemed redundant since no distinct advantages over the HLLC and Roe-Pike solvers are reported by Toro [108]. An assessment of the accuracy and robustness of an exact Riemann solver and the HLL, HLLC and Roe-Pike is included in section(4.1.1). Approximate Riemann solvers resolve a simplified wave structure by omitting the presence of intermediate waves (HLL, HLLC) or transforming the system of equations into a linearised form (Roe). The approximate wave structure of the HLL and HLLC approximate Riemann solvers is schematically illustrated in figure(3.5). These Riemann solvers have been incorporated and modified accordingly to ensure compatibility with the relative velocity formulation of the governing equations and are presented briefly in the next paragraphs.

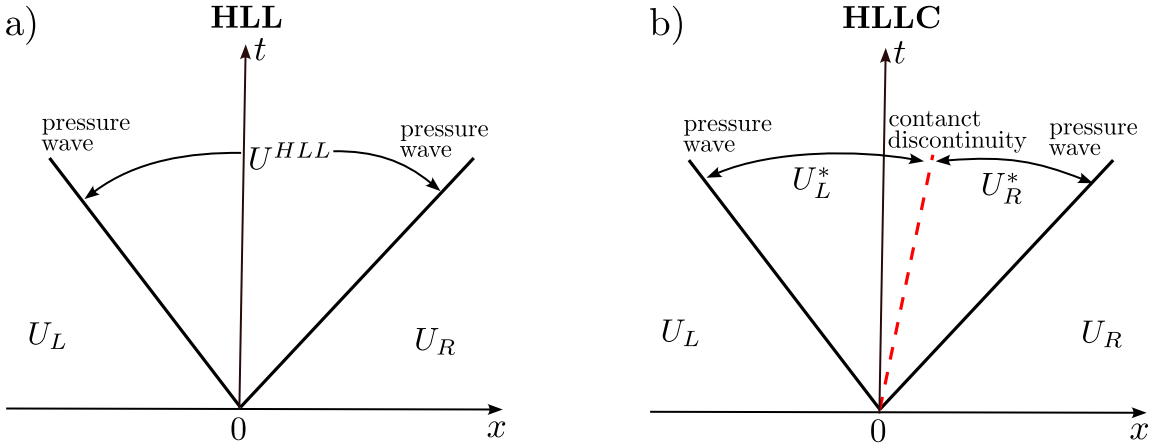


Figure 3.5: Wave structure of a) HLL approximate Riemann solver and b) HLLC Riemann problem (adapted from [108]).

The velocity notation used hereafter conforms with the velocity notation used by Toro [108] but does not reflect on the actual velocity components used in the solver. In practice, the local Riemann problem and the corresponding fluxes are evaluated along a local coordinate system, rotated by an angle ϕ_f around the circumferential axis. The axes of the local coordinate system are parallel and normal to the cell face vector, while the θ -axis of the global and the local coordinate system coincide. The fluxes are essentially evaluated in the streamwise m_f , spanwise n_f and circumferential

u_θ direction and the conformity between the conserved variable and flux nomenclature conventions is shown in eqns.(3.44) and (3.45).

The conservative variable vector conventions conform with each other as follows:

$$\mathbf{CV} = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E_0 \end{bmatrix} \sim \begin{bmatrix} \rho \\ \rho u_{m,f} \\ \rho u_\theta \\ \rho u_{n,f} \\ \rho E_0 \end{bmatrix} \quad (3.44)$$

The flux vector conventions conform with each other as follows:

$$\mathbf{F} = \begin{bmatrix} \rho u \\ \rho u^2 + P \\ \rho uv \\ \rho uw \\ \rho u H_0 \end{bmatrix} \sim \begin{bmatrix} \rho u_{m,f} \\ \rho u_{m,f}^2 + P \\ \rho u_{m,f} u_\theta \\ \rho u_{m,f} u_r \\ \rho u_{m,f} H_0 \end{bmatrix} \quad (3.45)$$

HLL Riemann solver

The HLL Riemann solver is an approximate Riemann solver which assumes a less complex wave structure, forming essentially a simplified two-wave model (see figure(3.5a)). The wave structure only consists of the two, leftmost and rightmost non-linear waves; the intermediate wave structure, consisting of 2 shear layers and one contact discontinuity, is omitted from the wave model. This results in inadequate resolution of features like contact surfaces, shear waves and material interfaces compared to both exact and other higher-resolution Riemann solvers such as HLLC and Roe's, compromising accuracy [108]. The two-wave model forms three distinct states, with the intermediate-star region being confined within the region in-between the two leftmost and rightmost waves and the left and right states being located left and right of the leftmost and rightmost waves respectively. The approximate HLL Riemann problem can be solved directly once the wave speeds are estimated; different algorithms for wave speed estimations have been proposed by different authors. The HLL scheme constitutes a simple, very robust and in a wide range of scenarios, a reasonably accurate approximate Riemann solver. The vectors of the primitive variables corresponding to the 3 states are given by:

$$\tilde{U}(x, t) = \begin{cases} \mathbf{U}_L & \text{if } \frac{x}{t} \leq S_L, \\ \mathbf{U}_{*L}^{HLL} & \text{if } S_L \leq \frac{x}{t} \leq S_*, \\ \mathbf{U}_R & \text{if } \frac{x}{t} \geq S_R, \end{cases} \quad (3.46)$$

Following the modelling assumptions of the HLL method, the flux vectors corresponding to the 3 constant states, are given by:

$$\mathbf{F}_{i+\frac{1}{2}}^{HLL} = \begin{cases} \mathbf{F}_L & \text{if } 0 \leq S_L, \\ \frac{S_R \mathbf{F}_L - S_L \mathbf{F}_R + S_L S_R (\mathbf{U}_R - \mathbf{U}_L)}{S_R - S_L} & \text{if } S_L \leq 0 \leq S_*, \\ \mathbf{F}_R & \text{if } 0 \geq S_R, \end{cases} \quad (3.47)$$

There are several approaches for the estimation of the speeds of the leftmost and rightmost waves, S_L and S_R and the evaluation of properties in the star region. A pressure-based wave estimation approach and a non-iterative scheme proposed by Toro [108], are used for the evaluation of the wave speeds and star properties. These schemes are discussed in more detail later.

HLLC Riemann solver

The HLLC constitutes an improved variant of HLL approximate Riemann solver, in the sense that it accounts for the presence of the contact discontinuity (see figure(3.5b)). Neither the HLL nor the HLLC solvers account for the presence of shear waves. The inclusion of the contact discontinuity dissects the single star region of the HLL method to a left and a right star region. The vectors of primitive variables and fluxes in each of the four regions are given by eqns.(3.48) and (3.49) respectively.

$$\tilde{U}(x, t) = \begin{cases} \mathbf{U}_L & \text{if } 0 \leq S_L, \\ \mathbf{U}_{*L} & \text{if } S_L \leq 0 \leq S_*, \\ \mathbf{U}_{*R} & \text{if } S_* \leq 0 \leq S_R, \\ \mathbf{U}_R & \text{if } 0 \geq S_R, \end{cases} \quad (3.48)$$

$$\mathbf{F}_{i+\frac{1}{2}}^{HLLC} = \begin{cases} \mathbf{F}_L & \text{if } 0 \leq S_L, \\ \mathbf{F}_{*L} & \text{if } S_L \leq 0 \leq S_*, \\ \mathbf{F}_{*R} & \text{if } S_* \leq 0 \leq S_R, \\ \mathbf{F}_R & \text{if } 0 \geq S_R, \end{cases} \quad (3.49)$$

At the intermediate states within the star region, the fluxes can be evaluated using 3 variants. The Riemann solver embedded in the code utilizes the first variant given by eqn.(3.50):

$$\mathbf{F}_{*K} = \mathbf{F}_K + S_K (\mathbf{U}_{*K} - \mathbf{U}_K) \quad (3.50)$$

The vector of conserved variables within the intermediate, star regions is given by eqn.(3.51):

$$\mathbf{U}_{*K}(x, t) = \rho_K \left(\frac{S_K - u_K}{S_K - S_*} \right) \begin{bmatrix} 1 \\ S_* \\ v_k \\ w_k \\ \frac{E_K}{\rho_K} + (S_* - u_K) \left[\frac{P_K}{\rho_K (S_K - u_K)} \right] \end{bmatrix} \quad (3.51)$$

Roe's Riemann solver - (Roe-Pike variant)

Roe's method replaces the Jacobian matrix by an approximate one, transforming the original non-linear system of equations into a linearised one with constant coefficients. The original Riemann problem is essentially replaced by an approximate, linearised one which is subsequently solved exactly. Roe-Pike method differentiates from the original method of Roe by evaluating the eigenvalues without the need of first defining the approximate Jacobian matrix. First the Roe average values are computed as follows:

$$\tilde{\rho} = \sqrt{\rho_L \rho_R} \quad (3.52)$$

$$\tilde{\mathbf{u}} = \frac{\sqrt{\rho_L \mathbf{u}_L} + \sqrt{\rho_R \mathbf{u}_R}}{\sqrt{\rho_L} + \sqrt{\rho_R}} \quad (3.53)$$

$$\tilde{\rho} = \frac{\sqrt{\rho_L H_L} + \sqrt{\rho_R H_R}}{\sqrt{\rho_L} + \sqrt{\rho_R}} \quad (3.54)$$

$$\tilde{\alpha} = \left((\gamma - 1) \left(\tilde{H} - \frac{1}{2} \tilde{V}^2 \right) \right) \quad (3.55)$$

The five eigenvalues of the linearised system of equations (one for each equation) are then given by:

$$\tilde{\lambda}_1 = \tilde{u} - \tilde{\alpha} \quad , \quad \tilde{\lambda}_2 = \tilde{\lambda}_3 = \tilde{\lambda}_4 = \tilde{u} \quad , \quad \tilde{\lambda}_5 = \tilde{u} - \tilde{\alpha} \quad (3.56)$$

The right eigenvectors are then computed as follows:

$$\tilde{\mathbf{K}}^1 = \begin{bmatrix} 1 \\ \tilde{u} - \tilde{\alpha} \\ \tilde{v} \\ \tilde{w} \\ \tilde{H} - \tilde{u}\tilde{\alpha} \end{bmatrix}, \tilde{\mathbf{K}}^2 = \begin{bmatrix} 1 \\ \tilde{u} \\ \tilde{v} \\ \tilde{w} \\ \frac{1}{2} \tilde{V}^2 \end{bmatrix}, \tilde{\mathbf{K}}^3 = \frac{1}{r} \frac{\partial}{\partial r} \begin{bmatrix} 0 \\ 0 \\ 1 \\ 0 \\ \tilde{v} \end{bmatrix}, \tilde{\mathbf{K}}^4 = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 1 \\ \tilde{w} \end{bmatrix}, \tilde{\mathbf{K}}^5 = \begin{bmatrix} 1 \\ \tilde{u}_x + \tilde{\alpha} \\ \tilde{u}_\theta \\ \tilde{u}_r \\ \tilde{H} + \tilde{u}\tilde{\alpha} \end{bmatrix} \quad (3.57)$$

The corresponding wave strengths are subsequently computed as follows:

$$\begin{bmatrix} \tilde{a}_1 \\ \tilde{a}_2 \\ \tilde{a}_3 \\ \tilde{a}_4 \\ \tilde{a}_5 \end{bmatrix} = \begin{bmatrix} \frac{1}{2\tilde{a}^2} [\Delta P - \tilde{\rho}\tilde{\alpha}\Delta u] \\ \Delta \rho - \frac{\Delta P}{\tilde{\alpha}^2} \\ \rho \Delta v \\ \rho \Delta w \\ \frac{1}{2\tilde{a}^2} [\Delta P + \tilde{\rho}\tilde{\alpha}\Delta u] \end{bmatrix} \quad (3.58)$$

Finally, the inter-cell flux of the Roe-Pike method is given by eqn.(3.59):

$$\mathbf{F}^{Roe} = \frac{1}{2} (\mathbf{F}_L + \mathbf{F}_R) - \frac{1}{2} \sum_{i=1}^m \tilde{a}_i \tilde{\lambda}_i \tilde{\mathbf{K}}^i \quad (3.59)$$

Estimation of wave speeds, S_L , S_R and pressure in the star region, P_*

The estimation of the pressure waves, S_L , S_R is a prerequisite for the determination of the HLL and HLLC inter-cell fluxes. Their exact value is not a priori known but can be estimated following either direct or pressure-based wave speed estimates [108]. Different approaches of the first variant have been successfully implemented; the simplest approach of Davis [111], estimates the velocities using minimum and maximum signal velocities of the left and the right states but is considered one of the least accurate and robust approaches. Roe's average eigenvalues have been utilised by Davis [111] and Einfeldt [112] to estimate the speed of the left and right non-linear waves, while Einfeldt [112] introduced later a modified version of the Roe-based wave speed estimations for the HLLE Riemann solver.

Pressure based, wave speed estimations, introduced by Toro [108], are an alternative, indirect wave speed estimation method, whereby wave speed estimations are a function also of the pressure inside the intermediate star region (P_*). This introduces an additional layer of complexity since P_* needs to be determined prior to wave-speeds. The pressure-based wave speed estimation approach recommended by Toro is implemented in the HLL and HLLC solvers. Depending on the type of the non-linear waves, the estimated wave speeds are computed using the exact, analytical equations corresponding to the speed of the head of the expansion fan or the speed of the shock wave, with the only approximation being the value of P_* . The wave speeds are computed as follows:

$$S_L = u_L - a_L q_L \quad \text{and} \quad S_R = u_R + a_R q_R \quad (3.60)$$

with

$$q_K = \begin{cases} 1 & \text{if } P_* \leq P_K \\ \sqrt{1 + \frac{\gamma+1}{2\gamma} \left(\frac{P_*}{P_K} - 1 \right)} & \text{if } P_* > P_K \end{cases} \quad (3.61)$$

The pressure and velocity terms inside the star region, P_* and u_* , are evaluated using the Adaptive Noniterative Riemann Solver (ANRS) proposed by Toro [108]. This combines three non-iterative schemes to solve for the star properties; a Primitive Variables Riemann Solver (PVRS) along with the Two-Rarefaction Riemann Solver (TRRS) and the Two-Shock Riemann Solver (TSRS). The principal idea behind

adaptive schemes is the implementation of more advanced, higher resolution Riemann solvers in regions dominated by highly non-uniform flows and large pressure gradients (including non-linear waves). The scheme switches to lower-order solvers in regions of uniform flow as well as near shear waves and contact discontinuities. The details of each individual Riemann solver are not presented here but can be found in [108].

Riemann solvers in the relative FoR

The Riemann solvers along with the conservative variable and flux vectors analysed in the previous sections, are expressed in the absolute FoR and naturally are not directly compatible when the relative velocity formulation is used. The conserved variable vector and the flux vector must be transformed into the equivalent relative ones, prior to the solution of the local Riemann problem in the rotating FoR. The conservative variable vector is initially transformed in the relative one, by replacing the absolute velocity vector with the relative one and the specific total energy, E_0 with the specific relative energy, E_r (eqn.(3.62)). The flux vector is transformed in the relative FoR by replacing the absolute velocity vector with the relative one and the stagnation enthalpy term with rothalpy (eqn.(3.63)). The conserved variable vector in the relative FoR is given by:

$$\mathbf{CV}_{abs} = \begin{bmatrix} \rho \\ \rho u_x \\ \rho u_\theta \\ \rho u_r \\ \rho E_0 \end{bmatrix} \Rightarrow \mathbf{CV}_{rel} = \begin{bmatrix} \rho \\ \rho w_x \\ \rho w_\theta \\ \rho w_r \\ \rho E_r \end{bmatrix} \quad (3.62)$$

,where the specific total energy, E_0 and specific relative energy, E_r , are given by eqns.(3.7) and (3.18) respectively. The fluxes are transformed in the rotating FoR as follows:

$$\mathbf{F}_{abs} = \begin{bmatrix} \rho u_x \\ \rho u_x^2 + P \\ \rho u_x u_\theta \\ \rho u_x u_r \\ \rho u_x H_0 \end{bmatrix} \Rightarrow \mathbf{F}_{rel} = \begin{bmatrix} \rho w_x \\ \rho w_x^2 + P \\ \rho w_x w_\theta \\ \rho w_x w_r \\ \rho w_x I \end{bmatrix} \quad (3.63)$$

,where the total enthalpy, H_0 and rothalpy, I , are given by eqns.(3.8) and (3.19).

3.2.8 Blockage terms evaluation

In sections(3.2.1) and (3.2.3) the conceptual modifications of the governing equations inside bladed sub-domains were presented. The reduction in the effective area due to metal and aerodynamic blockage phenomena is modelled by the introduction of a blockage parameter, b . Additional blockage gradient terms arise, representing the pressure forces exerted on the blocked regions. The precise evaluation of the blockage gradient term, $P\nabla b$, is not to trivial and erroneous or slightly inaccurate estimations precipitates significant momentum imbalance.

The evaluation of the blockage gradient terms using finite difference methods was deemed inadequate, due to the complex geometry of the centrifugal compressor ducts and the inaccurate evaluation of the terms, primarily at the LE and the TE. Errors stemming from inaccurate evaluation of those terms at the LE, were also mentioned by Kiwada [38] who adopted a finite difference approach to evaluate these terms in an axial compressor duct. An alternative, implicit, blockage gradient evaluation method was developed, capable of providing exceptionally accurate estimations in highly convoluted duct geometries; the blockage gradient terms are directly computed using the axial and radial momentum equations. A single pseudo-timestep is performed, temporarily setting the boundary conditions to atmospheric, i.e. $P_{0,in} = 101325Pa$, $T_{0,in} = 288.15K$ and $P_{out} = 101325Pa$, while the initial values correspond to stagnant air at atmospheric conditions, i.e. $u_x = w_x = 0m/s$, $u_{theta} = w_{theta} = 0m/s$, $u_r = w_r = 0m/s$ and $\rho = 1.225kg/m^3$. The full, transient solution of the flow-field in time is redundant; the flow-field should remain invariable and the steady-state solution of the flow-field should be identical to the initial one. The conserved variables in the first and the next time step are consequently identical, $CV^n = CV^{n+1}$ and the blockage gradient terms can be directly obtained by solving the axial and radial momentum equations for $P\partial b/\partial x$ and $P\partial b/\partial r$ respectively. Assuming, zero angular velocity along the flow-path and no body forces, eqn.(3.21) is reduced to the following form:

$$\frac{\partial}{\partial t}(b\rho\mathbf{u}) + \nabla \cdot (b\rho\mathbf{u}\mathbf{u} + bP) = P\nabla b \quad (3.64)$$

Due to initial and boundary conditions, the state of the flow-field is independent of time, i.e. $\frac{\partial}{\partial t} = 0$ and eqn. (3.64) can be further simplified as follows:

$$\nabla \cdot (b\rho\mathbf{u}\mathbf{u} + bP) = P\nabla b \quad (3.65)$$

The circumferential blockage gradient is zero since blockage is independent of the circumferential position ($\partial b / \partial \theta = 0$). The axial and radial momentum equations can be expanded using eqn.(3.65) and solved for the axial and radial blockage gradient terms as per eqns.(3.66) and (3.67):

Axial blockage gradient:

$$\frac{\partial b}{\partial x} = \frac{1}{P} \left[\nabla \cdot (b\rho u_x \mathbf{u}) + \frac{\partial(bP)}{\partial x} \right] \quad (3.66)$$

Radial blockage gradient:

$$\frac{\partial b}{\partial r} = \frac{1}{P} \left[\nabla \cdot (b\rho u_r \mathbf{u}) + \frac{\partial(bP)}{\partial r} - b\frac{P}{r} \right] \quad (3.67)$$

3.2.9 Discretisation

The governing equations are solved numerically using the finite volume method; the equations are time and volume-integrated over discrete finite volumes (cells) and the volume integrals are transformed to surface integrals using Gauss' divergence theorem. If $\mathbf{\Lambda}$ is a differentiable vector field, then the volume integrals can be transformed to surface integrals using eqn.(3.68):

$$\oint_A (\mathbf{\Lambda} \cdot \hat{\mathbf{n}}) dA = \iiint_V (\nabla \cdot \mathbf{\Lambda}) dV \quad (3.68)$$

The solution of the flow-field is obtained transiently using Godunov's scheme; inter-volume fluxes are obtained by solving the local Riemann problem at cell surfaces while forward time-stepping is used for the discretisation of the time-dependent terms. The blockage gradient terms are evaluated following the method discussed in the

previous section, while the artificial viscosity terms are evaluated separately, using finite difference schemes. The discretised governing equations in the relative FoR with blockage, body forces and artificial diffusion terms in matrix form are given by eqn.(3.69).

$$\begin{aligned} \mathbf{C}\mathbf{V}^{n+1} = \mathbf{C}\mathbf{V}^n + \frac{\Delta t}{\Delta V} [b_{i,j}A_{i,j}\mathbf{F}_{i,j} - b_{i+1,j}A_{i+1,j}\mathbf{F}_{i+1,j} + b_{i,j}A_{i,j}\mathbf{G}_{i,j} - b_{i,j+1}A_{i,j+1}\mathbf{G}_{i,j+1}]^n \\ + \rho\Delta t \left[\mathbf{f}_{cyl} + \mathbf{f}_{rot} + \mathbf{f}_{\tau} + \mathbf{f}_{bl} + \mathbf{B}\frac{1}{b_{vol}} \right]_{i,j}^n \end{aligned} \quad (3.69)$$

In eqn.(3.69), the superscript, n , denotes the time step, with n expressing the current time step. The streamwise and spanwise indices of the cells are denoted with i and j respectively. A schematic describing the grid index nomenclature is shown in figure(3.6).

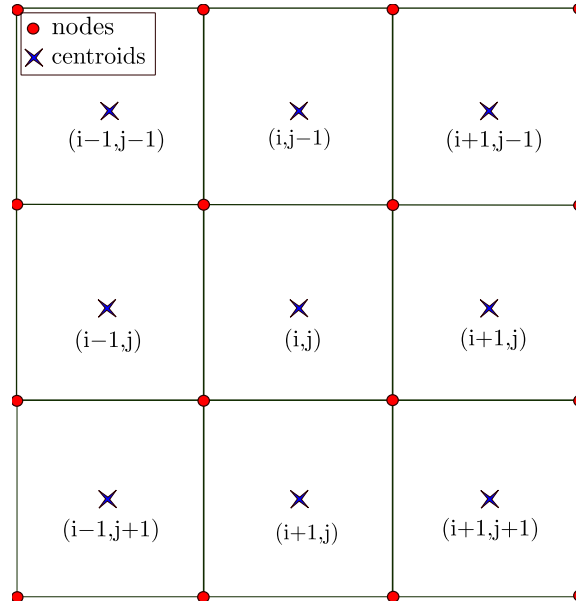


Figure 3.6: Index nomenclature along computational grid.

In eqn.(3.69), $\mathbf{C}\mathbf{V}$ is the conserved variable vector, $\mathbf{F}$ the streamwise flux vector, $\mathbf{G}$ the spanwise flux vector, $\mathbf{f}_{cyl}$ the body force source terms due to the formulation of the equations in the cylindrical coordinate system, $\mathbf{f}_{rot}$ the Coriolis and centrifugal acceleration body force source terms, $\mathbf{f}_{\tau}$ the shear stress terms, $\mathbf{f}_{bl}$ the blade body forces and $\mathbf{B}$ the blockage source terms. The conserved variable and flux vectors in the streamwise and spanwise direction are given by eqn.(3.70).

$$\mathbf{C}\mathbf{V}_{abs} = \begin{bmatrix} b\rho \\ b\rho w_{m,stream} \\ b\rho w_\theta \\ b\rho w_{n,stream} \\ b\rho E_{rel} \end{bmatrix}, \quad \mathbf{F} = \begin{bmatrix} b\rho w_{m,stream} \\ b\rho w_{m,stream}^2 + P \\ b\rho w_\theta w_{m,stream} \\ b\rho w_{n,stream} w_{m,stream} \\ b\rho w_{m,stream} I \end{bmatrix}, \quad \mathbf{G} = \begin{bmatrix} b\rho w_{m,span} \\ b\rho w_{m,span}^2 + P \\ b\rho w_\theta w_{m,span} \\ b\rho w_{n,span} w_{m,span} \\ b\rho w_{m,span} I \end{bmatrix} \quad (3.70)$$

The body force source terms along with the artificial diffusion and blockage term vectors are given by eqn.(3.71):

$$\mathbf{f}_{cyl} = \begin{bmatrix} 0 \\ 0 \\ \frac{b\rho w_\theta w_r}{r} \\ -\left(\frac{b\rho w_\theta^2}{r} + \frac{bP}{r}\right) \\ 0I \end{bmatrix}, \quad \mathbf{f}_{rot} = \begin{bmatrix} 0 \\ 0 \\ 2b\rho\Omega w_r \\ -(2b\rho\Omega w_\theta + b\rho\Omega^2 r) \\ 0 \end{bmatrix}, \quad \mathbf{f}_{bl} = \begin{bmatrix} 0 \\ b\rho f_x \\ b\rho f_\theta \\ b\rho f_r \\ 0 \end{bmatrix},$$

$$\mathbf{f}_\tau = \begin{bmatrix} 0 \\ b\rho\nu^* \left(\cos\phi_f \frac{\partial^2 V_{m,f}}{\partial n_f^2} - \sin\phi_f \frac{\partial^2 V_{n,f}}{\partial m_f^2} \right) \\ 0 \\ b\rho\nu^* \left(\sin\phi_f \frac{\partial^2 V_{m,f}}{\partial n_f^2} + \cos\phi_f \frac{\partial^2 V_{n,f}}{\partial m_f^2} \right) \\ b\rho\nu^* \left[\left(\frac{\partial u_{m,f}}{\partial n_f} + \frac{\partial u_{n,f}}{\partial m_f} \right)^2 + u_{m,f} \frac{\partial^2 u_{m,f}}{\partial n_f^2} + u_{n,f} \frac{\partial^2 u_{n,f}}{\partial m_f^2} \right] \end{bmatrix}, \quad \mathbf{B} = \begin{bmatrix} 0 \\ P \frac{\partial b}{\partial x} \\ 0 \\ P \frac{\partial b}{\partial r} \\ 0 \end{bmatrix} \quad (3.71)$$

Finite difference approximations - artificial diffusion terms

The numerical solution of the artificial diffusion terms presented in section(3.2.6) is based on a simpler, finite difference approach. Central finite differencing is used, thus the approximation of the terms is second order accurate; this allows for higher order resolution of those terms in a numerically efficient way. It should be noted

that these terms are purely meant to provide a mixing mechanism and promote, to a certain extent, momentum and energy transfer in the streamwise and spanwise direction, thus a more sophisticated numerical evaluation method was not deemed necessary. At the domain boundaries, where no upstream or downstream centroids are available, fictitious centroids with identical flow properties to those of boundary centroids are assumed; this locally reduces the accuracy to first order but linearly extrapolated values could be used instead to restore the accuracy of the scheme. If u denotes a velocity component, then the first order and second order derivatives with respect to the streamwise and spanwise direction can be approximated as follows:

First order derivative approximations - Forward FD:

$$\left. \frac{\partial u}{\partial m} \right|_i = \frac{u_{i+1} - u_i}{\Delta m} + O(\Delta m) \quad (3.72)$$

First order derivative approximations - Backward FD

$$\left. \frac{\partial u}{\partial m} \right|_i = \frac{u_i - u_{i-1}}{\Delta m} + O(\Delta m) \quad (3.73)$$

First order derivative approximations - Central FD

$$\left. \frac{\partial u}{\partial m} \right|_i = \frac{u_{i+1} - u_{i-1}}{2\Delta m} + O(\Delta m)^2 \quad (3.74)$$

Second order single derivative approximations - Central FD

$$\left. \frac{\partial^2 u}{\partial m^2} \right|_i = \frac{u_{i+1} - 2u_i + u_{i-1}}{(\Delta m)^2} + O(\Delta m)^2 \quad (3.75)$$

Second order mixed derivative approximations - Central FD

$$\left. \frac{\partial^2 u}{\partial m \partial n} \right|_i = \frac{u_{i+1,j+1} - u_{i-1,j+1} - u_{i+1,j-1} + u_{i-1,j-1}}{4\Delta m \Delta n} + O[(\Delta m)^2, (\Delta n)^2] \quad (3.76)$$

The finite difference approximations given by eqns.(3.72) - (3.76) are strictly valid on uniform grids. The discretised flow-path domain incorporates a combination of uniform subdomain grids adjoined by variable node distribution grids. The uniform-grid, finite difference approximations are also implemented in the non-uniform sections of the domain since the small, gradual changes in the streamwise dimensions of the cells are not expected to introduce any noteworthy errors. No errors associated with the uniform grid limitations of eqns.(3.72) - (3.76) are introduced when gradients are evaluated in the spanwise direction since the spanwise length of the cells at every streamwise position is constant.

3.3 Blade metal blockage and curvature definition

The local evaluation of the turning force is dependent on the local curvature distribution of the blade geometry. The full blade curvature in three dimensions can be fully determined via the definition of three angles; blade metal angle, lean angle and flow-path angle. The blade metal angle, κ , expresses the curvature of the blade camberline with respect to the meridional direction, essentially dictating the turning of the flow due to the curvature of the camberline. Rotor blades usually deflect the flow in the opposite direction to that of rotation, therefore rotor blade angles in the code are typically assigned a negative sign while stator angles a positive. The adopted sign convention does not exclude the possibility of local regions with opposite sign angles being present (for example stage 67). The lean angle, λ , expresses the leaning of the blade in the circumferential direction, with a positive angle designating blade lean towards the axis of rotation. The flow path angle, ϕ , describes the geometric curvature of the duct along the meridional plane, with $\phi = 0^\circ$ representing a fully axial duct and $\phi = 90^\circ$ a fully radial one. The reduction in the effective annulus area due to the finite thickness of the blades is expressed by the blockage parameter, b . The blockage parameter indicates spatially the amount of the circumferential extent which is not blocked, i.e. $b = 0.8$ indicates that locally, 80% of the circumferential extent is unobstructed or conversely, 20% of the annulus is blocked. The camber surface grid, the definition of the blade angles along with the corresponding unit vectors are schematically presented in figure(3.7).

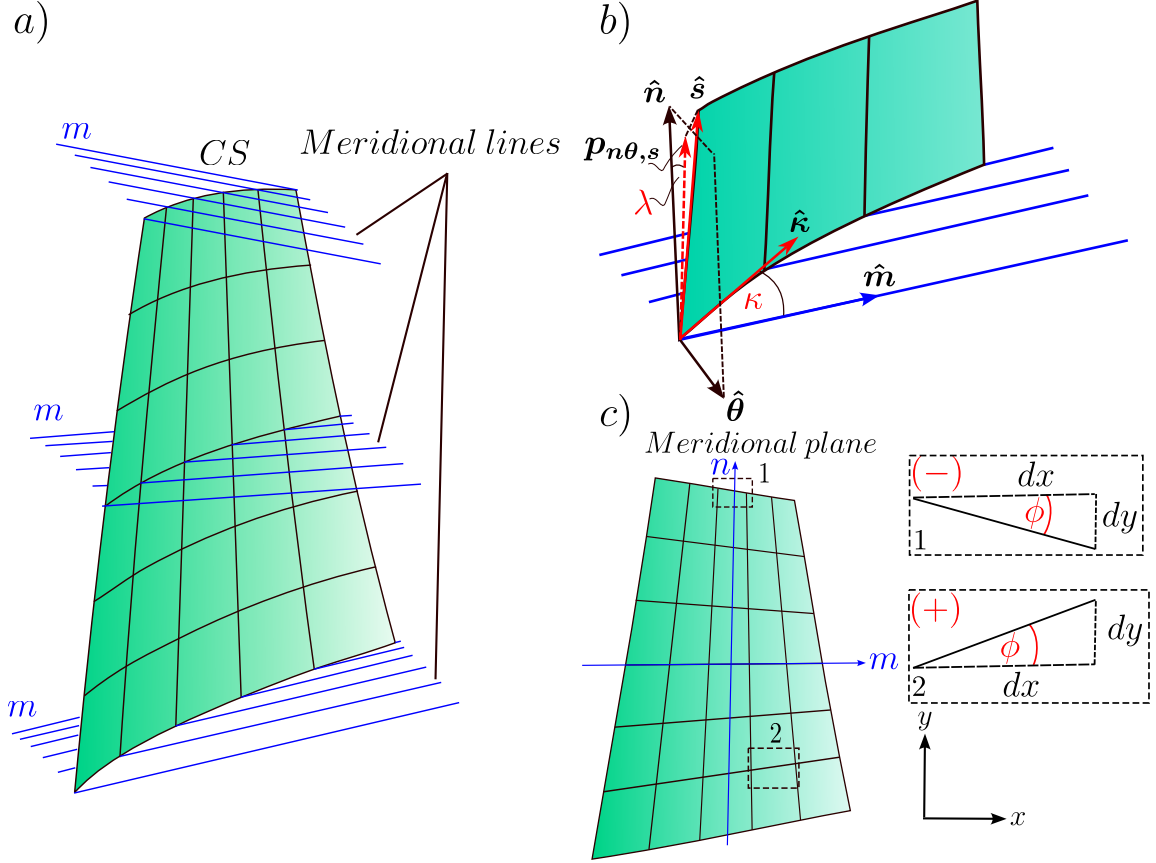


Figure 3.7: a) Camber surface grid and meridional lines passing through the nodes of the grid for the estimation of blade curvature. Definition of the b) blade and lean angle and c) flow-path angle (defined in the meridional plane).

The local distribution of the curvature and the blockage is performed numerically, along the vertices forming the surface meshes of the camber, pressure and suction side of each blade. The camber surface mesh is used to obtain the angle distributions, while the pressure and the suction meshes to estimate the local blade blockage. The camber surface is generated by taking the arithmetic average of the (x, θ, r) coordinates of the pressure and suction side (eqn.(3.77)).

$$\begin{aligned}
 x_{CS} &= \frac{x_{PS} + x_{SS}}{2} \\
 \theta_{CS} &= \frac{\theta_{PS} + \theta_{SS}}{2} \\
 r_{CS} &= \frac{r_{PS} + r_{SS}}{2}
 \end{aligned} \tag{3.77}$$

The extraction of the blade angles is performed on individual nodes via a vector-based analysis by defining local unit vectors aligned with the local blade geometry and the coordinates forming the planes of interest. The blade metal angle is derived by the dot product of two vectors; the local camber line vector and the local meridional vector. The camber line vector, $\hat{\mathbf{k}}$, is aligned with the local camberline of the blade at different span positions. The local meridional vector, $\hat{\mathbf{m}}$, is the vector aligned with the meridional lines, rotated by the corresponding θ angle to ensure that the meridional lines pass through each vertex. The blade angle is estimated using eqn.(3.78), where the $(-)$ is introduced to ensure compatibility with the aforementioned blade angle sign convention.

$$\kappa = -\text{acos}\left(\frac{\hat{\mathbf{m}} \cdot \hat{\mathbf{k}}}{|\hat{\mathbf{m}}||\hat{\mathbf{k}}|}\right) \quad (3.78)$$

The lean angle, λ , is measured along the (n, θ) plane, where $\hat{\mathbf{n}}$ is the vector normal to the meridional, $\hat{\mathbf{m}}$ and circumferential, $\hat{\boldsymbol{\theta}}$ unit vectors. The lean angle is defined as the angle between two unit vectors; the $\hat{\mathbf{n}}$ unit vector and the projection of the vector parallel to the blade surface grid lines in the hub-to-tip direction, $\hat{\mathbf{s}}$, on the plane $(n - \theta)$ denoted by $\mathbf{p}_{n\theta,s}$. The unit vector $\hat{\mathbf{s}}$ essentially is the vector parallel to the path formed by the vertices in the spanwise direction. The projection of the $\hat{\mathbf{s}}$ unit vector on the $(n - \theta)$ plane is given by eqn.(3.79):

$$\mathbf{p}_{n\theta,s} = \hat{\mathbf{s}} - \frac{\hat{\mathbf{s}} \cdot \hat{\mathbf{m}}}{|\hat{\mathbf{m}}|^2} \hat{\mathbf{m}} \quad (3.79)$$

The lean angle is then the angle between the $\hat{\mathbf{n}}$ and the $\mathbf{p}_{n\theta,s}$ vectors and is computed using the dot product of the two vectors as follows:

$$\lambda = \text{acos}\left(\frac{\mathbf{p}_{n\theta,s} \cdot \hat{\mathbf{n}}}{|\mathbf{p}_{n\theta,s}||\hat{\mathbf{n}}|}\right) \quad (3.80)$$

The flow path angle is defined as the angle between the meridional and the axial vector along the meridional plane. It is defined using the coordinates of adjacent vertices:

$$\phi = \text{atan}\left(\frac{dr}{dx}\right) \quad (3.81)$$

The blade metal blockage is locally computed using the coordinates of the pressure and suction side surfaces generated using piecewise interpolation and not the raw blade data. This ensures a similar/equivalent node distribution on both surfaces since the generation of the blade's surface grids are generated using the same interpolation

scheme and number of elements in both the streamwise and the spanwise direction. The thickness of the blade at each (x, θ, r) position is given by:

$$t_{bl} = (\theta_{PS} - \theta_{SS}) \cdot \frac{r_{PS} + r_{SS}}{2} \quad (3.82)$$

The blockage parameter designates at every (m, n) position the fraction of the circumferential extent which is not blocked due to the presence of blades and/or aerodynamic blockage. In the presence of splitter blades, their blade metal blockage distribution is independently evaluated and subsequently added to the blockage introduced by the main blades. The blockage parameter is given by:

$$b = 1 - \left[\frac{N_{fb} t_{fb}}{\pi(r_{PS,fb} + r_{SS,fb})} + \frac{N_{sb} t_{sb}}{\pi(r_{PS,sb} + r_{SS,sb})} \right] \quad (3.83)$$

,where the subscripts fb and sb refer to full blade and splitter blade quantities respectively. A schematic overview illustrating the camber, pressure and suction surface meshes and the definition of the blade blockage parameter are given in figure(3.8).

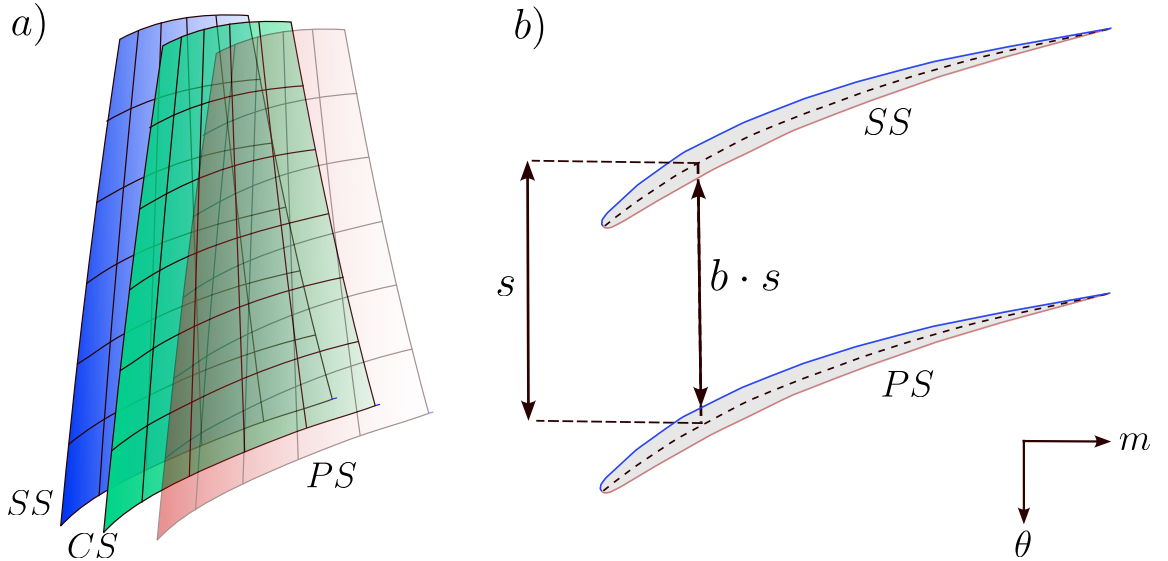


Figure 3.8: a) Pressure, suction and camber surface mesh of an axial rotor blade and b) definition of blockage parameter.

3.3.1 Blade metal blockage and curvature interpolation

The blade angles are externally computed on blade surface meshes, thus the surface grid elements do not coincide with the elements of the main computational domain. To transfer the computed blade angle and blockage distributions from the camber surface grid onto the main computational grid, interpolation is required. The blade curvature and blockage data stored in the nodes of the camber surface mesh are projected onto the meridional plane. Bilinear interpolation is performed by defining arbitrary interpolating quadrilaterals enclosing the centroids of the main grid. The arbitrary quadrilaterals are not formed in the (x, r) but rather the natural (m, n) coordinate system and are subsequently converted to unit squares using mapping functions as shown in figure(3.9).

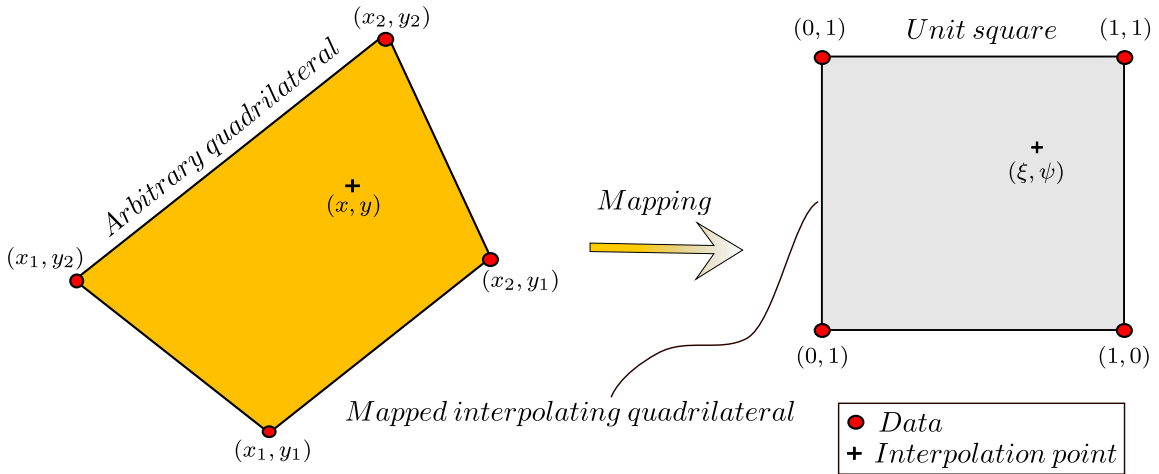


Figure 3.9: Transformation of arbitrary interpolating quadrilateral to a unit square quadrilateral.

The transformation of the arbitrary interpolating quadrilaterals, defined by 4 points in the (x, y) coordinates system, to an equivalent unit square (ξ, ψ) is performed via the following mapping functions:

$$x = \alpha_1 + \alpha_2\xi + \alpha_3\psi + \alpha_4\xi\psi \quad (3.84)$$

$$y = \beta_1 + \beta_2\xi + \beta_3\psi + \beta_4\xi\psi$$

,where the coefficients α_i and β_i are given by eqn.(3.85):

$$\begin{aligned}
\boldsymbol{\alpha} &= \begin{bmatrix} 1 & 0 & 0 & 0 \\ -1 & 1 & 0 & 0 \\ -1 & 0 & 0 & 1 \\ 1 & -1 & 1 & -1 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ x_3 \\ x_4 \end{bmatrix} \\
\boldsymbol{\beta} &= \begin{bmatrix} 1 & 0 & 0 & 0 \\ -1 & 1 & 0 & 0 \\ -1 & 0 & 0 & 1 \\ 1 & -1 & 1 & -1 \end{bmatrix} \begin{bmatrix} y_1 \\ y_2 \\ y_3 \\ y_4 \end{bmatrix}
\end{aligned} \tag{3.85}$$

The transformed coordinates (ξ, ψ) are defined so that $(\xi_1, \psi_1) = (0, 0)$, $(\xi_2, \psi_2) = (1, 0)$, $(\xi_3, \psi_3) = (1, 1)$ and $(\xi_4, \psi_4) = (0, 1)$. The ξ coordinate is computed by eqn.(3.86).

$$\xi = \frac{x - \alpha_1 - \alpha_3 m}{\alpha_2 + \alpha_4 m} \tag{3.86}$$

The ψ coordinate is computed by solving the quadratic function as shown in eqn.(3.87).

$$\begin{aligned}
&\overbrace{(\alpha_4 \beta_3 - \alpha_3 \beta_4)}^a \psi^2 + \overbrace{(\alpha_4 \beta_1 - \alpha_1 \beta_4 + \alpha_2 \beta_3 - \alpha_3 \beta_2 + x \beta_4 - y \alpha_4)}^b \psi \\
&\quad + \overbrace{(\alpha_2 \beta_1 - \alpha_1 \beta_2 + x \beta_2 - y \alpha_2)}^c = 0
\end{aligned} \tag{3.87}$$

$$\psi = \frac{-b + \sqrt{b^2 - 4ac}}{2a}$$

The interpolated property is subsequently estimated as a weighted average of the values stored in the 4 nodes of the interpolating quadrilateral enclosing the centroid on the main grid. Assuming that the blade angles on the camber surface mesh nodes, κ_{CS} , are known, the interpolated blade metal angles, κ , are approximated by:

$$\kappa = (1 - \xi)(1 - \psi)\kappa_{CS}|_1 + \xi(1 - \psi)\kappa_{CS}|_2 + \xi\psi\kappa_{CS}|_3 + (1 - \xi)\psi\kappa_{CS}|_4 \tag{3.88}$$

3.4 Body force model

The entire compressor is modelled as an empty, 2D-axisymmetric duct, thus no distinct blade grids are present along the flow path. The meridional projection of the blades' leading and trailing edges define regions where body force-fields are exerted. The effect of turbomachinery components on the local flow-field is threefold; in the rotating FoR, the turning of the flow is primarily governed by the local camber angle of the blade. The local aerofoil camberline imposes a change in the local relative streamline curvature. This change is modelled by introducing an isentropic *turning force*, acting perpendicular to the relative velocity vector. The second effect is viscous dissipation. This manifests as a loss in the rotary stagnation pressure, or equivalently a reduction in the absolute or relative stagnation pressure relative to the corresponding isentropic ones. Entropy generation is strictly associated with forces acting parallel and against the relative flow path [76, 91]. An analytical *viscous force* model coupled with empirical loss correlations are combined to predict and impose an equivalent viscous force. The last effect is the energy exchange whereby in stationary components energy is zero since no energy is added, while in rotating domains and depending on the type of the rotating component, energy is either added (compressor) or extracted from the flow (turbine). This energy exchange is only perceivable in the absolute FoR.

3.4.1 Turning force model

The turning of the flow and the corresponding turning force is a function of the local flow-field variables and the local curvature of the imposed relative path. Considering a compressor blade, the curvature of the ideal imposed relative path coincides with that of the local camberline of the blade, expressed by the local blade metal angle, κ . Viscous phenomena such as boundary layer development along the blade profile cause the relative streamlines to diverge from the *ideal camberline* path by a local deviation angle, δ . Local changes to the relative path with respect to the camberline might also be the outcome of inviscid flow mechanisms; flow slip in centrifugal compressors is an inviscid-induced phenomenon causing the flow to deviate from the blade local camberline [16]. To account for flow deviation, an *effective camberline* may be defined, the curvature of which, is expressed by the imposed relative flow angle, which is the summation of the local blade angle and the deviation angle, $\beta_{imp} = \kappa + \delta$. Since the *turning force* is contingent upon the imposed local curvature of the imposed relative path, its evaluation at every timestep requires that

the deviation angle is known a priori. The deviation angle may then be extracted directly from CFD simulations or estimated by means of component-specific empirical loss correlations. In the first approach, local deviation angles along the blade domain can be extracted and the imposed relative flow path can be explicitly defined at all cells inside blade domains. In the latter, a function is required in order to distribute the predicted deviation angle in the blade passage.

The *turning force* acts perpendicular to the relative path (which includes the local deviation angle) thus no entropy generation is associated with the turning of the flow. To ensure the applicability of the model irrespective of the blade type under consideration e.g. axial rotor/stator, impeller blade, vaned diffuser blade etc., the blade curvature needs first to be fully defined. The curvature of the blade in space is defined using three angles discussed in section(3.3); the blade metal angle, the flow path angle and the lean angle. Figure(3.10) illustrates the forces acting on a fluid particle in the relative FoR, in the direction normal to the relative streamline, annotated with ν . The forces comprise the components of the pressure gradient, the Coriolis and centripetal accelerations and the forces due to the formulation of the equations in the cylindrical coordinate system.

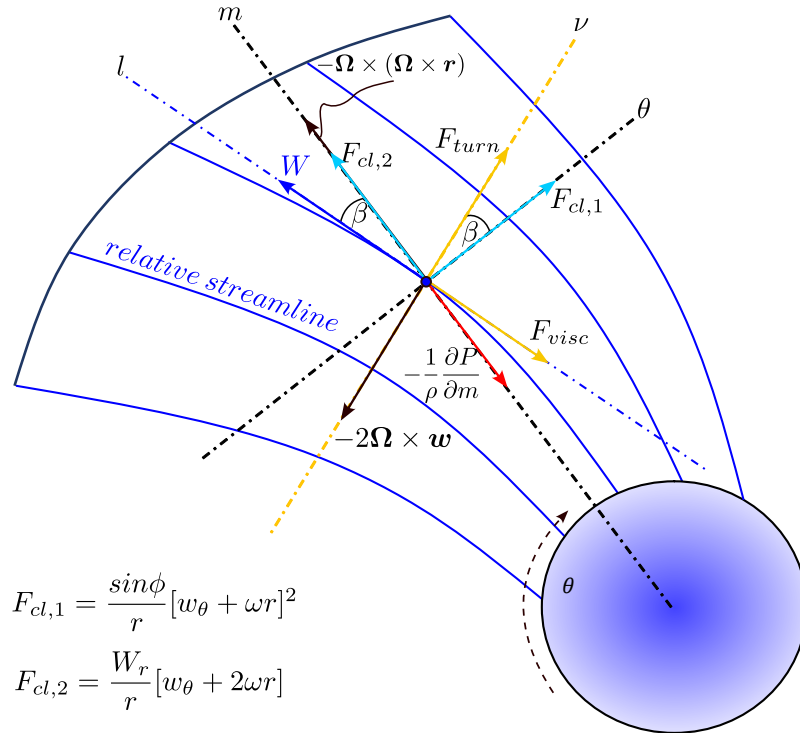


Figure 3.10: Body force balance along a relative streamline for the derivation of the body force model.

The turning force is a function of both the local curvature of the blade in three dimensions as well as the local flow-field variables. The basis of the turning force model is derived by considering momentum balance along a relative streamline using the Lagrangian form of the Euler equations and steady-state conditions ($\partial/\partial t = 0$). Along the duct, the body force field acts on every finite volume positioned within the geometrical boundaries of a blade. Since no distinct blade passages are modelled, the body forces act as if an infinite number of blades were guiding the flow, thus the circumferential fluxes inside an infinitely thin blade passage are zero ($\partial/\partial \vartheta = 0$). The angles of compressor rotors are considered negative since they deflect the flow circumferentially, in the direction opposite to that of rotation (which is considered positive). The inviscid momentum equations in vector form are given by eqn.(3.89).

$$(\mathbf{w} \cdot \nabla) \mathbf{w} = -\frac{1}{\rho} \nabla P - 2\boldsymbol{\Omega} \times \mathbf{w} - \boldsymbol{\Omega} \times (\boldsymbol{\Omega} \times \mathbf{r}) \quad (3.89)$$

The left hand side can be expanded as follows [102]:

$$(\mathbf{w} \cdot \nabla) \mathbf{w} = w \left(\frac{\partial w}{\partial l} \hat{\mathbf{l}} + \frac{\partial \hat{\mathbf{l}}}{\partial l} w \right) = w \left(\frac{\partial w}{\partial l} \hat{\mathbf{l}} - \frac{\partial \beta}{\partial l} \hat{\boldsymbol{\nu}} \right) \quad (3.90)$$

The component of the momentum term in the normal to the relative path direction is $-w \frac{\partial \beta}{\partial l}$ and is a function of the relative velocity and the relative flow angle gradient with respect to the direction parallel to the relative flow path. The RHS is expanded by projecting the blade body forces, Coriolis, centrifugal and the additional source terms due to the formulation of the equations in cylindrical coordinates, in the direction normal to the flow path. Omitting in the analysis the lean angle and solving for the turning force, the force balance in the direction normal to the relative streamline (annotated with ν in figure(3.10)), gives eqn.(3.91) which essentially is a reduced form of Kottapalli's turning force model [85].

$$F_{turn} = -\frac{1}{\rho} \frac{\partial P}{\partial n} - w^2 \left(\frac{\partial \beta}{\partial l} + \frac{\sin \beta \sin \phi}{r} \right) + 2\Omega w \sin \phi + \Omega^2 r \sin \beta \sin \phi \quad (3.91)$$

As discussed in section(2.4.1), the presence of a pressure gradient term in eqn.(3.91) is associated with instabilities [85, 88] and is expected to be particularly unstable at the presence of large local pressure gradients. An influence coefficient analysis [102] similar to Brand [88] and Kottapalli [85] is performed to express the normal-to-the-flow-path pressure gradient term, $\partial P/\partial n$, as a function of other flow-field variables. The derivation of the term, as a function of other flow-field variables is performed

by combining the differential form of 6 equations; continuity, meridional momentum, Gibb's equation, Marble's viscous force expression, conservation of rothalpy and equation of state. The derivation is thoroughly presented in appendix(6.4).

Equation of state:

$$\frac{dP}{P} = \frac{d\rho}{\rho} + \frac{dT}{T} \quad (3.92)$$

Meridional area:

$$\frac{dA_m}{A_m} = \frac{dr}{r} \quad (3.93)$$

Continuity:

$$\frac{d\rho}{\rho} + \frac{A_m}{A_m} + \frac{dw}{w} + \tan\beta d\beta = 0 \quad (3.94)$$

Gibb's equation:

$$\frac{\nabla s}{c_p} = \frac{\nabla T}{T} - \frac{\gamma - 1}{\gamma} \frac{\nabla P}{P} \quad (3.95)$$

Viscous force using Marble's expression:

$$F_{visc} = \frac{\partial s}{\partial l} T = \frac{\partial s}{\partial m} T \cos\beta \quad (3.96)$$

Conservation of rothalpy:

$$\frac{dT}{T} = \frac{dw}{w} \left[-(\gamma - 1) Ma_{rel}^2 \right] + \frac{dr}{r} \left[(\gamma - 1) Ma_{\Omega}^2 \right] \quad (3.97)$$

Equations (3.92) - (3.4.1) are combined to express the normal-to-the-relative-flow-path gradient $\partial P/\partial n$ as a function of other flow-field variables. The pressure gradient term $\partial P/\partial n$, expressed as a function of other flow-field parameters is given by eqn(3.98). The influence coefficient analysis and the derivation eqns.(3.92)–((3.98)) is included in appendix(6.4).

$$\begin{aligned} \frac{1}{\rho} \frac{dP}{dn} = \frac{w^2}{(2\gamma - 1) Ma_{rel}^2 + 1} & \left\{ -F_{visc} \tan\beta \frac{\frac{1}{Ma_{rel}^2} + (\gamma - 1)}{R(\gamma - 1)} \right. \\ & \left. + \frac{\sin\phi \sin\beta}{r} \left(\frac{Ma_{\Omega}^2}{Ma_{rel}^2} + 1 \right) - \sin\beta \tan\beta \frac{\partial\beta}{\partial m} \right\} \end{aligned} \quad (3.98)$$

Substituting eqn.(3.98) into eqn.(3.91), the final form of the new turning force model is derived (eqn.(3.99)).

$$\begin{aligned}
F_{turn} = w^2 & \left[\frac{\cos\beta ((2\gamma - 1) Ma_{rel}^2 - \tan^2\beta + 1)}{(2\gamma - 1) Ma_{rel}^2 + 1} \frac{\partial\beta}{\partial m} + \frac{\frac{Ma_{\Omega}^2}{Ma_{rel}^2} + (2\gamma - 1) Ma_{rel}^2 + 2}{(2\gamma - 1) Ma_{rel}^2 + 1} \frac{\sin\phi \sin\beta}{r} \right] \\
& + F_{visc} \left[-\frac{\tan\beta \left(\frac{1}{Ma_{rel}^2} + (\gamma - 1) \right)}{(\gamma - 1) R} \right] + 2w\Omega [\sin\phi] + \Omega^2 r [\sin\phi \sin\beta]
\end{aligned} \tag{3.99}$$

The first term constitutes the component due to streamline curvature along both the $(\vec{x} - \vec{r})$ and the $(\vec{m} - \vec{\theta})$ planes expressed by the flow path angle, ϕ and relative flow angles, β respectively. The second term designates the viscous force contribution, while the last two terms encapsulate the contribution of the Coriolis and centrifugal acceleration due to the formulation of the turning force model in the rotating FoR.

3.4.2 Correction force model

The turning force model provides the core component of the turning force required to locally align the flow with the imposed flow direction. Inside a time-dependent flow-field, local flow-field variables may be subject to substantial gradients, especially when source terms (e.g. body forces) are present or highly time-dependent phenomena are simulated (e.g rotating stall and surge). The presence of these gradients may result to local misalignment of the flow, severely compromising the ability of the turning force model to accurately impose the correct amount of flow turning. The accumulation of these local misalignment errors leads to a numerically-induced increase or decrease in the imposed deviation angle and consequently in the predicted amount of energy exchange along the blade row.

To ensure that during both transient and steady-state conditions, the prescribed amount of flow turning is accurately imposed, an additional force component is introduced, which will be referred to as *correction force*. The coupling of a main turning force model with an additional correction force component is reported in other through-flow, body-force-based, compressor modelling approaches, focusing on simulating rotating stall and surge in axial compressors [27, 86, 91] as discussed in section(2.4.1). The flow misalignment correction introduced by these models was

found inadequate for impeller and vaned diffuser blades, mainly due to the radius-dependent characteristic length (pitch length) used in these models. The pitch length of impeller and vaned diffuser blades substantially decreases from the LE to the TE due to the large change in radius. This leads to a substantial reduction in the magnitude of the applied correction force especially in the vicinity of the TE, which can severely compromise the ability of the model to successfully impose the local relative flow angle. Flow misalignment at the TE entails substantial implications on global compressor aerodynamic performance predictions, since the amount of overall flow turning, dictates the energy input into the flow.

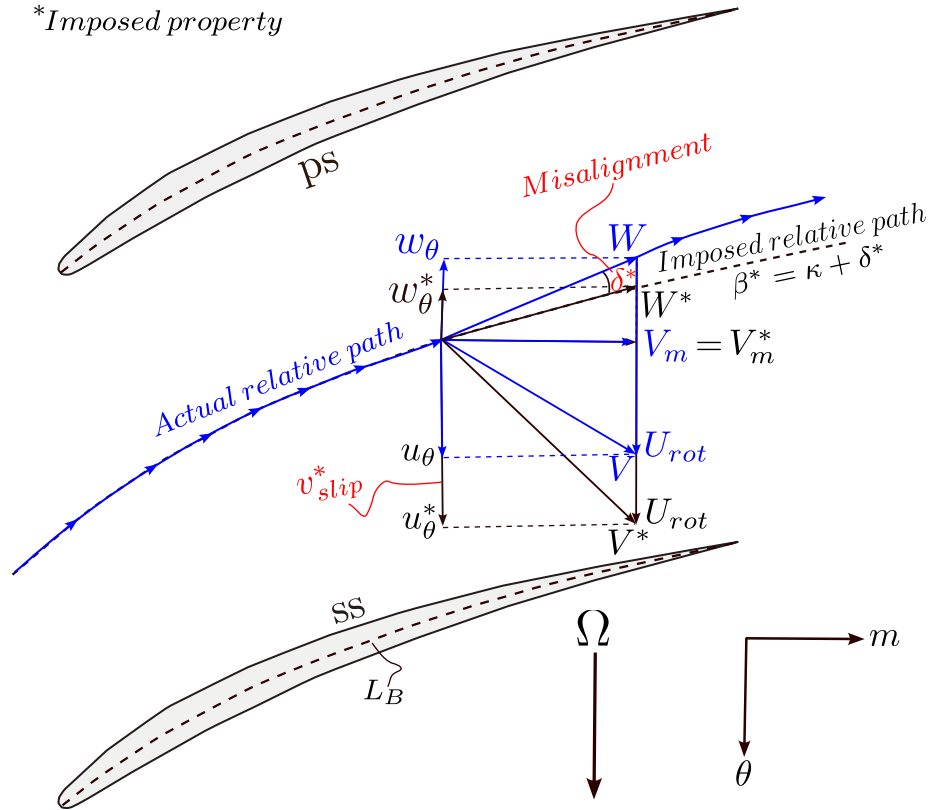


Figure 3.11: Overview of correction force model velocity components due to flow misalignment with respect to the imposed relative flow path.

The new correction force model is formulated, following the concept of numerical slip, while a more adequate, radius-independent characteristic length is used. The magnitude of the applied correction force is adjusted in a transient manner via a numerical slip factor coefficient (eqn.(3.100)). The numerical slip factor is computed in the absolute FoR, assuming that the tangential velocity components differ due to flow slippage, while the meridional velocity components of the ideal, perfectly guided flow are identical (figure(3.100)). The meridional velocity component dictates the mass flow and is not considered to vary due to over or under-turning of the flow.

This numerical slippage of the flow is isentropic since it is purely associated with the over- or underestimation of the required turning force magnitude. The corresponding numerical slip velocity is given by eqn.(3.102).

$$\sigma_{cor} = \frac{u_{\theta,act}}{u_{\theta,imp}} = \frac{u_{\theta,act} + \Omega r}{u_{\theta,imp} + \Omega r} \quad (3.100)$$

Numerical slip purely manifests as a change in the actual absolute circumferential velocity component, $u_{\theta,act}$ and consequently the absolute velocity component, V_{act} compared to the velocity components corresponding to a perfectly guided flow (denoted with *). It follows that the relative circumferential velocity component, $w_{\theta,act}$ and the corresponding relative velocity magnitude, W_{act} will also differ due to the misalignment of the flow with respect to the imposed relative flow path (figure(3.11)). Since the meridional velocity is assumed to be independent of flow slip, the circumferential velocity components in eqn.(3.100) can be expressed as follows:

$$\sigma_{cor} = \frac{u_{\theta,act}}{u_{\theta,imp}} = \frac{V_{mer} \tan \beta_{act} + \Omega r}{V_{mer} \tan \beta_{imp} + \Omega r} \quad (3.101)$$

The numerical slip velocity, $u_{slip,cor}$ due to the misestimation of turning force can be expressed as:

$$u_{slip,cor} = u_{\theta,act} - u_{\theta,imp} = u_{\theta,act} - u_{\theta,act} \left[\frac{u_{\theta,imp}}{u_{\theta,act}} \right] = u_{\theta,act} \left[\frac{\sigma_{cor} - 1}{\sigma_{cor}} \right] \quad (3.102)$$

To constrain the ability of the correction force model's to drastically interfere and dominate the amount of turning imposed to the flow which could lead to numerical instabilities, the correction force is expressed as a function of the correction slip velocity, $u_{slip,cor}$, the meridional velocity, V_{mer} and a user-defined coefficient, K_{cor} . A characteristic length is introduced to ensure unit consistency; the local camberline or geometric meridional streamline length were found to be the most adequate characteristic lengths. Contrary to the blade pitch, which was incorporated in other correction force models, camberline and geometric meridional streamline lengths are independent of the large pitch length changes which in centrifugal compressors are particularly more pronounced towards the TE. The correction force coefficient, K_{cor} , essentially constitutes a tuning parameter, with higher values introducing larger correction forces and more accurate flow guidance. Very high K_{cor} values tend to increase the "stiffness" of the flow and may lead to numerical instabilities and unrealistic flow behaviour. The correction force model is given by eqn.(3.103):

$$F_{cor} = K_{cor} \frac{u_{slip,cor} |V_{mer}|}{L_{stream}} = K_{cor} \frac{u_{\theta,act} |V_{mer}|}{L_{stream}} \left[\frac{\sigma_{cor} - 1}{\sigma_{cor}} \right] \quad (3.103)$$

The correction force model (eqn.(3.103)) in expanded form is given by:

$$F_{cor} = K_{cor} \frac{|V_{mer}| u_{\theta,act}}{L_{stream}} \left[1 - \frac{V_{mer} \tan \beta_{imp} + \Omega r}{u_{\theta,act}} \right] \quad (3.104)$$

,or using relative velocity components:

$$F_{cor} = K_{cor} \frac{|V_{mer}| (w_{\theta,act} + \Omega r)}{L_{stream}} \left[1 - \frac{V_{mer} \tan \beta_{imp} + \Omega r}{w_{\theta,act} + \Omega r} \right] \quad (3.105)$$

Since the correction force model is applied in the relative FoR, an equivalent, less complicated and equally efficient reduced version of the correction force model, given by eqns.(3.103) - (3.105), can be derived by simply omitting the angular velocity terms, i.e. setting $\Omega = 0$. In that formulation, the numerical slip factor, σ_{cor} reduces to:

$$\sigma_{cor} = \frac{w_{\theta,act}}{w_{\theta,imp}} = \frac{V_{mer} \tan \beta_{act}}{V_{mer} \tan \beta_{imp}} \quad (3.106)$$

The alternative correction force model in the relative FoR is given by eqn.(3.107):

$$F_{cor} = K_{cor} \frac{|V_{mer}| w_{\theta,act}}{L_{stream}} \left[1 - \frac{V_{mer} \tan \beta_{imp}}{w_{\theta,act}} \right] \quad (3.107)$$

3.4.3 Viscous force model

To model the viscous dissipation associated with the non-isentropic phenomena in blade passages (profile losses, tip clearance losses etc.), a viscous body force source term is introduced. The *viscous force* is always acting parallel to the relative flow path and opposite to the direction of motion, thereby constituting by definition an entropy generation source term. This is evident by eqn.(3.110) according to which any body force acting in the direction parallel and opposite to the relative streamline generates entropy. An application of (eqn.(3.110)) inside an inviscid flow-field is not directly possible unless the entropy gradient along the relative path is known a priori e.g. by means of 3D single passage RANS simulations. This would require an extensive CFD-extracted database of the entropy gradient at different flow conditions for all modelled

blade rows. This limitation is addressed in the current model by incorporating *loss correlations* which transiently predict the pressure losses based on inlet and outlet geometric and flow-field parameters. This enables universal applicability of the viscous force model to different blade designs and flow conditions, ultimately imparting predictive capabilities to it. The correlations account for different sources of entropy generation mechanisms originating from mixing, skin friction, tip clearance etc. The cumulative effect of those loss mechanisms is introduced into the flow via the viscous force and losses are distributed in the spanwise and streamwise direction via *distribution coefficients* (see section(3.4.4)).

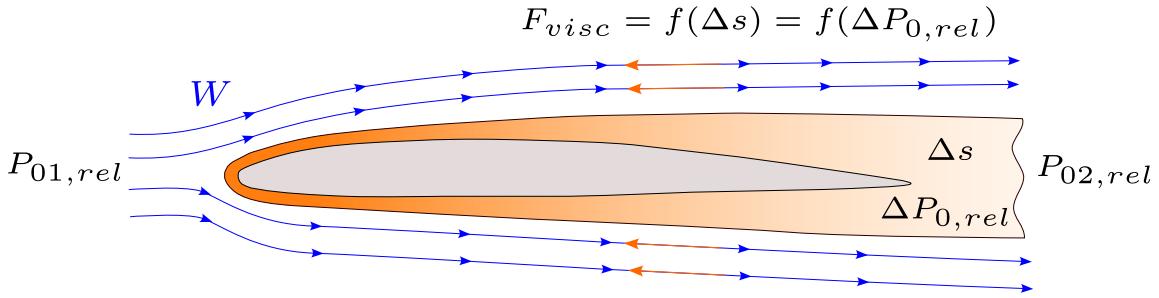


Figure 3.12: Viscous force acting parallel to the relative streamlines generating entropy and stagnation pressure loss.

The new analytical viscous force model relates the viscous force with the total pressure losses in the relative FoR. The derivation is performed in appendix(6.4) by combining the differential form of rothalpy and Gibbs' equation along with Marble's viscous force expression eqn.(3.4.1).

Gibbs' equation using relative stagnation properties:

$$T_{0,rel}ds = dh_{0,rel} = -\frac{1}{\rho}dP_{0,rel} \quad (3.108)$$

Rothalpy:

$$dI = dh_{0,rel} - \frac{1}{2}d(\Omega^2 r^2) \quad (3.109)$$

Viscous force using Marble's expression:

$$\mathbf{F}_{visc} = \frac{\partial s}{\partial l} T \hat{\mathbf{l}} = \frac{\partial s}{\partial m} T \cos \beta \hat{\mathbf{l}} \quad (3.110)$$

Flow path angle:

$$\sin \phi = \frac{dr}{dm} \quad (3.111)$$

Combining eqns.(3.108) - (3.4.1), the final form of the new viscous force model is derived and expressed by eqn.(3.112).

$$\mathbf{F}_{visc} = \cos \beta \frac{P}{\rho} \left[\frac{1}{P_{0,rel}} \frac{\partial P_{0,rel}}{\partial m} - \frac{\sin \phi \gamma}{\left(1 + \frac{\gamma - 1}{2} Ma_{rel}^2\right)} \frac{Ma_{\Omega}^2}{r} \right] \hat{\mathbf{l}} \quad (3.112)$$

Equation(3.112) relates the viscous force to the meridional relative stagnation pressure gradient as well as other flow-field variables. The first term in the brackets represents the changes in the relative stagnation pressure due to both isentropic-related effects, as well as viscous phenomena. The first, stem from changes in blade velocity, $u_{rot} = \Omega r$ which along a non-accelerating blade row are exclusively related to changes in the geometric flow path radius. The latter encapsulate entropy generation which manifests as a reduction in the relative stagnation pressure with respect to the ideal one. The second term balances out changes in the relative stagnation pressure gradient which are the outcome of strictly isentropic-related phenomena e.g. changes in u_{rot} . The meridional pressure gradient term can be decomposed into a numerically more convenient way as follows:

$$\frac{\partial P_{0,rel}}{\partial m} = \frac{\partial P_{0,rel}}{\partial m} \Big|_{id} - \frac{\partial P_{0,rel}}{\partial m} \Big|_{loss} \approx \frac{\Delta P_{0,rel}|_{id} - \Delta P_{0,rel}|_{act}}{\Delta m} \quad (3.113)$$

The origins of the $\Delta P_{0,rel}|_{id}$ can be identified by defining stagnation properties in the rotating FoR often referred to as rotary stagnation properties [113]. Inside an inviscid rotating domain, it is the rotary stagnation pressure and rotary stagnation temperature that remain constant; relative stagnation quantities remain constant only

if the radius of the relative streamline is constant e.g. inviscid flow in a fully axial rotor. Any change in the radius of the relative streamline inside a rotating domain will result in a change of the relative stagnation pressure even in a fully isentropic flow. The rotary stagnation temperature (eqn.(3.115)) is derived by rothalpy, while rotary stagnation pressure (eqn.(3.116)) is expressed using isentropic relations either in the absolute or relative FoR.

$$I = h_{0,rel} - \frac{1}{2}U^2 \quad (3.114)$$

$$T_{0,\Omega} = T_{0,rel} - \frac{U^2}{2C_p} \quad (3.115)$$

$$P_{0,\Omega} = P_{0,rel} \left(1 - \frac{\frac{U^2}{2C_p}}{T_{0,rel}} \right) \quad (3.116)$$

Rotary stagnation temperature is always conserved in an adiabatic flow without energy addition, while rotary stagnation pressure is conserved only if the flow is strictly isentropic. If the rotational speed is constant along a relative streamline, then changes in rotary stagnation properties are identical to relative ones. This is also expressed by the viscous force model; the second term in the brackets represents the relative stagnation enthalpy which in an adiabatic flow with no energy addition is identical to the isentropic one. Any imbalance, by means of relative stagnation pressure loss, will generate entropy. The second term is also proportional to the flow-path angle ϕ , thus in a fully axial domain entropy generation is proportional to the relative stagnation pressure loss gradient along the relative path.

As discussed earlier, the evaluation of $dP_{0,rel}$ requires an a priori knowledge of both the inviscid total pressure change $dP_{0i,rel}$ and the viscous-induced total pressure change $dP_{0loss,rel}$ in the relative FoR. Setting the meridional entropy gradient to zero and solving eqn.(3.112) for $dP_{0,rel}$, the inviscid relative stagnation pressure $dP_{0i,rel}$ can be estimated by eqn.(3.117). Alternatively, isentropic relations can be used. The local, viscous-induced $dP_{0loss,rel}$ is distributed along the relative path by multiplying the predicted average passage pressure loss (predicted by loss correlations) with a modified viscous force distribution coefficient, $C_{visc}^*(m, n)$ to account for the fact that the estimated relative stagnation pressure loss is not the average but rather the aggregate pressure loss along the cascade. The imposed change in the relative stagnation pressure along the relative streamline is the difference between the ideal

component and the locally distributed pressure loss, expressed as per eqn.(3.118).

$$dP_{0i,rel} \approx \Delta P_{0i,rel} = \left(\frac{\gamma \sin \phi}{1 + \frac{\gamma - 1}{2} Ma_{rel}^2} \right) \frac{Ma_{\Omega}^2}{r} P_{0,rel} \Delta m \quad (3.117)$$

$$dP_{0,rel} \approx \Delta P_{0,rel} = \Delta P_{0,rel}|_{id} - C_{visc}^*(m, n) \overline{\Delta P_{0,rel}}|_{loss} \quad (3.118)$$

Using the distribution coefficient concept, only an average, relative stagnation pressure loss of the entire blade passage is required, which is transiently obtained by loss correlations. The embedded loss correlations models are specific for the type of blade row under consideration and include axial rotor blades, IGVs, impellers and vaned diffuser blades. The distribution coefficient can be either a user-defined function or can be derived from CFD simulations (see section (3.4.4)). The viscous force is approximated at each individual cell using eqn.(3.119)

$$\mathbf{F}_{visc} \approx \cos \beta \frac{P}{\rho} \left[\frac{1}{P_{0,rel}} \frac{\Delta P_{0,rel}|_{id} - C_{visc}^*(m, n) \overline{\Delta P_{0,rel}}|_{loss}}{\Delta m} - \frac{\sin \phi \gamma}{\left(1 + \frac{\gamma - 1}{2} Ma_{rel}^2\right)} \frac{Ma_{\Omega}^2}{r} \right] \hat{\mathbf{t}} \quad (3.119)$$

3.4.4 Distribution coefficients

The *distribution coefficient* approach aims to provide a mathematical description of the local distribution of any flow-field variable under consideration. The influence of the distribution coefficients on the body-force model is two-pronged; to simplify the evaluation of certain terms of both the turning and viscous force models and to provide a means of distributing those variables either by user-defined functions or by more complex but physically realistic CFD-derived distributions.

CFD-extracted distributions provide a more sophisticated and realistic mapping approach for the distribution of any flow-field variable of interest since the coefficients are derived using data directly obtained from 3D, single-passage, RANS simulations. CFD-derived coefficients are capable of capturing and reconstructing the losses and the local flow misalignment due to complex, viscous-induced such as boundary layers, tip leakage vortices and separations. If the average value of the flow-field variables are known, then the flow-field can be reconstructed by expressing these variables as a function of the corresponding coefficients. User-defined distribution functions are incapable of reproducing such complex flow features, however they

offer a simplified CFD-independent modelling approach that provides control over the imposed distribution, providing enhanced stability and robustness. In certain occasions, regions of large viscous forces e.g. at the tip, may locally distribute very large viscous forces which could result in an unrestrained, naturally-occurring flow separation or flow-reversal. This could compromise the ability of the model to impose the computed averaged deviation computed by the correlations.

A distribution coefficient is defined as the ratio of the local to the average value of a variable along a direction, plane or volume (eqn.(3.120)). The average value of a consistently defined coefficient whether custom or CFD-derived, along a specified direction, area or volume should be equal to unity, $\overline{C}_\Lambda = 1$. Assuming that a flow-solution along the meridional plane of a blade row is available, distribution coefficients in the streamwise and spanwise direction can be defined according to eqns.(3.121) and (3.122). The integration limits of eqn.(3.123) would then coincide with the stream position of the LE and TE, while the integration limits of eqn.(3.124) would coincide with the span position of the hub and the shroud.

$$C_\Lambda = \frac{\Lambda}{\overline{\Lambda}} \quad (3.120)$$

$$C_{\Lambda,m}(m) = \frac{\Lambda(m)}{\overline{\Lambda}_{stream}} \quad (3.121)$$

$$C_{\Lambda,n}(n) = \frac{\Lambda(n)}{\overline{\Lambda}_{span}} \quad (3.122)$$

,where the streamwise and spanwise averaged properties $\overline{\Lambda}_{stream}$, $\overline{\Lambda}_{span}$ are evaluated from eqns(3.123) and (3.124) respectively.

$$\overline{\Lambda}_{stream} = \frac{\int_{m_1}^{m_2} \Lambda(m) dm}{\Delta m} \quad (3.123)$$

$$\overline{\Lambda}_{span} = \frac{\int_{n_1}^{n_2} \Lambda(n) dn}{\Delta n} \quad (3.124)$$

Equations (3.121) and (3.122) describe the distribution coefficient of a quantity $\Lambda(m, n)$ along the streamwise and the spanwise coordinate respectively with the intrinsic assumption that $C_{\Lambda,m}(m)$ and $C_{\Lambda,n}(n)$ are independent of n, m directions respectively. An analysis adopting the aforementioned assumption would be highly inaccurate since both the streamwise and spanwise distributions vary substantially along the span and stream direction. A valid approach would be to derive $C_{\Lambda,m}(m)$

and $C_{\Lambda,n}(n)$ at different (n, m) positions respectively but this would impose implementation constraints due to the large amount of required data. Reasonably accurate approximate distribution coefficients may be obtained by averaging multiple local $C(m)_{\Lambda,m}$ and $C(n)_{\Lambda,n}$ at different n, m positions. It is then possible to reduce the local $C_{\Lambda,m}(m)$ and $C_{\Lambda,n}(n)$ coefficients to a single pair of averaged $\overline{C(m)}_{\Lambda,m}$, $\overline{C(n)}_{\Lambda,n}$ coefficients. The averaging process provides representative distributions of the whole domain but local flow features that significantly differ from the average trend e.g. a very high local deviation angle at the shroud, might be suppressed or smoothed out due to the averaging process.

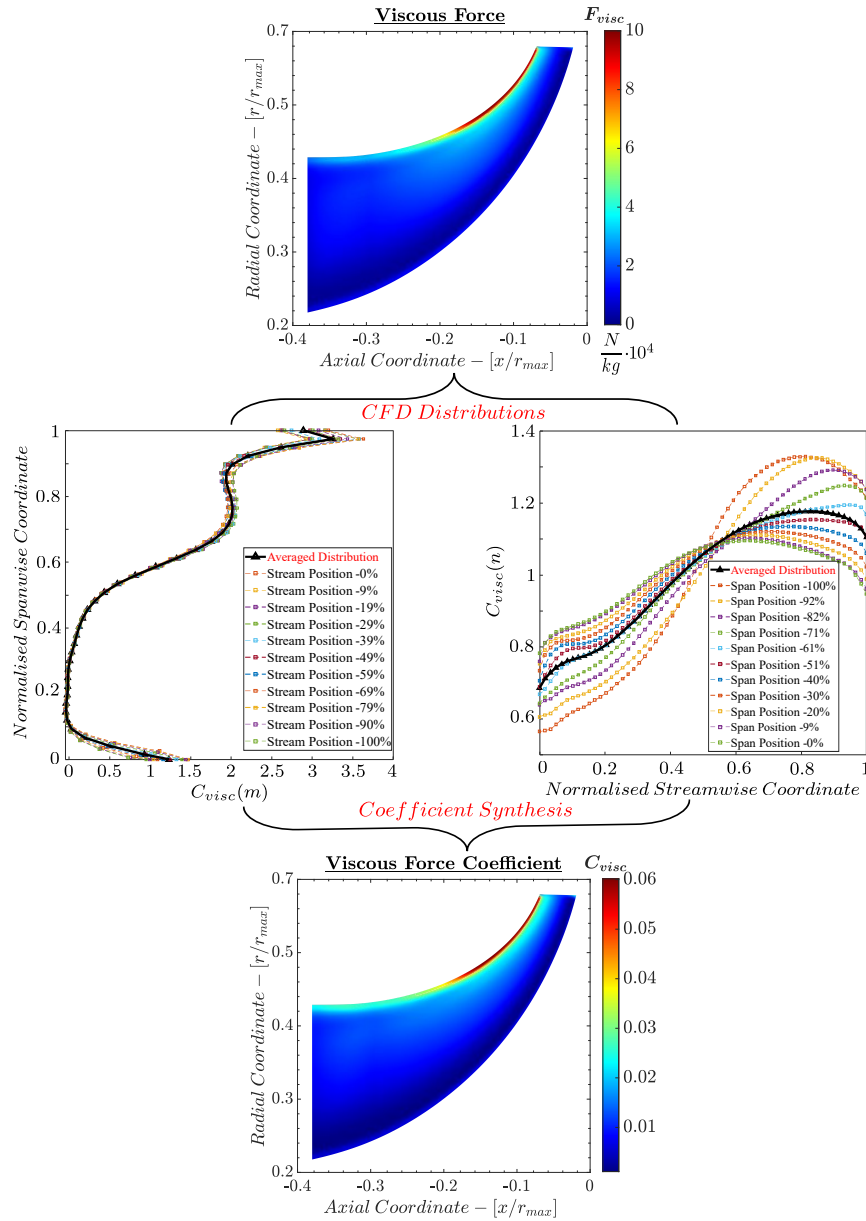


Figure 3.13: Extraction and synthesis process of viscous force distribution coefficients using averaged streamwise and spanwise distribution profiles extracted from CFD simulations.

Figure(3.13) illustrates the extraction and synthesis process of a viscous force distribution coefficient from CFD results, using averaged streamwise and spanwise distribution coefficients at different locations in the blade passage. Despite the averaging process, the coefficient reconstructs the viscous force with remarkable accuracy, preserving the high local viscous forces at the shroud. This method was found to be less accurate for the extraction of deviation distribution due to higher non-uniformity of the deviation angle distribution. The averaged streamwise and spanwise distribution coefficients may be combined into a single total distribution coefficient $\overline{C_{total}}(m, n)$ to mathematically describe the full distribution of any variable Λ along the meridional plane. The merging of the averaged streamwise and spanwise distributions is mathematically described by eqn.(3.125). This method was used for the extraction of the coefficient shown in figure(3.13). Alternatively, coefficients can be derived by utilizing the entire dataset, effectively normalizing the data with an area, mass-flow or volume-averaged property, $\bar{\Lambda}$, as per eqn.(3.126). The distribution coefficient is subsequently interpolated on the centroids of the computation grid.

$$\overline{C_{\Lambda, total}}(m, n) = \overline{C_{\Lambda, n}}(n) \frac{\int_{n_1}^{n_2} \overline{C_{\Lambda, m}}(m) dn}{\Delta n} \quad (3.125)$$

$$C_{\Lambda, total}(m, n) = \frac{\Lambda(m, n)}{\bar{\Lambda}} \quad (3.126)$$

3.4.5 Body force decomposition

Inside bladed domains, body-forces are exerted on finite volume cells to turn the flow and introduce entropy generation to account for viscous-induced phenomena that cannot be reproduced due to the inviscid nature of the governing equations. The first is acting perpendicular to the relative streamline, while the latter parallel and opposite to the direction of motion. The decomposition of the forces in the cylindrical coordinate system (r, θ, x) provides the source terms introduced in the axial, circumferential and radial momentum equations. The local coordinate system, aligned with the relative streamline and the blade surface, can be described by the unit vectors $(\hat{\mathbf{k}}, \hat{\mathbf{v}}, \hat{\mathbf{l}})$, where $\hat{\mathbf{l}}$ the unit vector parallel to the streamline (and the viscous force), $\hat{\mathbf{v}}$ the unit vector normal to the relative streamline and normal to the blade surface and $\hat{\mathbf{k}}$ the unit vector perpendicular to $\hat{\mathbf{l}}$ and $\hat{\mathbf{v}}$.

Turning force decomposition

The turning force is exerted parallel to the direction designated by the unit vector $\hat{\nu}$, so $\mathbf{F}_{turn} = F_{turn}\hat{\nu}$. Initially, the turning force is projected on the (k, h, m) coordinate system, which results from the rotation of the (k, ν, l) coordinate system around the ν - axis by the relative flow angle, β :

$$\begin{aligned} F_{k,turn} &= 0 \\ F_{h,turn} &= F_{turn}\cos\beta \\ F_{m,turn} &= F_{turn}\sin\beta \end{aligned} \tag{3.127}$$

To account for the effect of the lean angle, the forces are subsequently projected on the coordinate system (n, θ, m) , which results from the rotation of the (k, h, m) system around the m axis, by the lean angle, λ .

$$\begin{aligned} F_{n,turn} &= -F_{turn}\cos\beta\sin\lambda \\ F_{\theta,turn} &= F_{turn}\cos\beta\cos\lambda \\ F_{m,turn} &= F_{turn}\sin\beta \end{aligned} \tag{3.128}$$

To account for the meridional curvature of the streamlines, expressed by the flow-path angle ϕ , the force components are decomposed on the cylindrical coordinate system (r, θ, x) by rotating the coordinate system (n, θ, m) around the θ axis, by the local flow path angle ϕ :

$$\begin{aligned} F_{r,turn} &= F_{turn} \{ \sin\beta\sin\phi - \cos\beta\sin\lambda\cos\phi \} \\ F_{\theta,turn} &= F_{turn} \{ \cos\beta\cos\lambda \} \\ F_{x,turn} &= F_{turn} \{ \sin\beta\cos\phi + \cos\beta\sin\lambda\sin\phi \} \end{aligned} \tag{3.129}$$

Viscous force decomposition

The viscous force is acting parallel to the direction defined by the unit vector $\hat{l}$, so $\mathbf{F}_{visc} = F_{turn}\hat{l}$. The viscous force is initially projected on the (k, h, m) coordinate system, which results from the rotation of the (k, ν, l) coordinate system around the ν - axis by the relative flow angle, β :

$$\begin{aligned}
F_{k,visc} &= 0 \\
F_{h,visc} &= F_{visc} \sin \beta \\
F_{m,visc} &= F_{visc} \cos \beta
\end{aligned} \tag{3.130}$$

To account for the effect of the lean angle, the forces are subsequently projected on the coordinate system (n, θ, m) , which results from the rotation of the (k, h, m) around the m - axis, by the lean angle, λ .

$$\begin{aligned}
F_{n,visc} &= -F_{visc} \sin \beta \sin \lambda \\
F_{\theta,visc} &= F_{visc} \sin \beta \cos \lambda \\
F_{m,visc} &= F_{visc} \cos \beta
\end{aligned} \tag{3.131}$$

The force components are finally decomposed on the cylindrical coordinate system (r, θ, x) by rotating the coordinate system (n, θ, m) around the θ axis by the local flow path angle, ϕ . A sign correction is applied due to the negative sign convention used for rotor blade angles. Assuming that the force of the viscous force is positive, the viscous force components on the cylindrical coordinate system are given by eqn.(3.132):

$$\begin{aligned}
F_{r,visc} &= F_{visc} \{ \cos \beta \sin \phi + \cos \phi \sin \lambda \sin \beta \} \\
F_{\theta,visc} &= -F_{visc} \{ \sin \beta \cos \lambda \} \\
F_{x,visc} &= F_{visc} \{ \cos \beta \cos \phi - \sin \phi \sin \lambda \sin \beta \}
\end{aligned} \tag{3.132}$$

Sign convention

Within the context of the developed body force model, blade metal angles of rotors are considered negative, since they typically deflect the flow in a direction opposite to that of rotation. The angular velocity of the blades is always considered positive; if the blade curvature of a rotor blade locally deflects the flow towards the direction of rotation, then the angle is considered positive, otherwise a negative sign is assigned. To ensure consistency with the sign convention, a negative sign is assigned to the relative flow angles in eqn.(3.129) and eqn.(3.132). The viscous force, F_{visc} , is by definition considered negative when the streamwise velocity is positive and positive when the streamwise velocity is negative, thereby ensuring that it constitutes an entropy-generation force regardless of the direction of the flow.

3.5 Blockage-mixing model

At reverse flow conditions, in the vicinity of the trailing edge, large regions of separated flow occupy large fractions of the blade passage. In axial rotors, the separation bubble was found to be rather uniform, occupying a substantial area of the blade passage and extending from the LE to the TE. This implies that most of the mixing process occurs downstream the LE, essentially once the flow exits the rotor. In larger blades, e.g. impeller and vaned diffuser blades, the higher residence time of the flow inside the blade passage, results in a much higher degree of mixing inside the passage. This results in smaller and less uniform, relative to the blade passage size, separation regions. These do not fully extend along the blade passage (from LE to TE) since the separated region progressively diminishes due to the mixing process. In both cases, mixing-out of the flow, results in additional mixing losses and changes in the direction of the flow downstream the LE. The development of a simplified, analytical model capable of estimating these phenomena was deemed necessary to improve global reverse flow compressor performance predictions as well as to ensure more physically-accurate, flow-field solutions.

3.5.1 Blockage model

The reverse flow aerodynamic blockage model is formed on the basis of the coexistence of two distinct flow streams with the first representing the bulk flow and the latter the blocked region due to the presence of a separation bubble. The model essentially is based on the experimental observations of Gamache and Greitzer [59], while the derivation was performed on the principal ideas of Longley's simplified blockage-mixing model [91]. The bulk/jet flow is considered to be perfectly guided by the blades and fully isentropic; momentum and energy transfer due to shear layers is considered negligible. To facilitate the derivation of the model, assumptions are required; the flow is considered incompressible while the separation bubble along the blade passage is considered invariable and uniform.

The aforementioned assumptions are simplifications that often do not realistically reflect on the actual flow physics. Frequently, large relative Mach numbers are encountered, mainly in high-speed compressors at large pressure ratios, thus compressibility effects often are not negligible in blade passages during reverse flow operation. The assumption of constant-size separation bubbles is reasonably accurate in axial compressor blades but not in long blades where substantial mixing occurs inside the passage. Flow mixing inside larger passages is considered in the analysis by

expressing the aerodynamic blockage as a function of a mixing parameter. Flow deviation in axial rotor blades and vaned diffusers at reverse flow conditions is typically insignificant, but depending on the state of the flow in the vicinity of the LE, larger deviation angles might be encountered due to the presence of recirculation regions and other complex flow patterns. A schematic overview, representative of an axial rotor/stator or impeller passage preceded by an entirely axial duct where mixing at constant area takes place, is illustrated in figure(3.14).

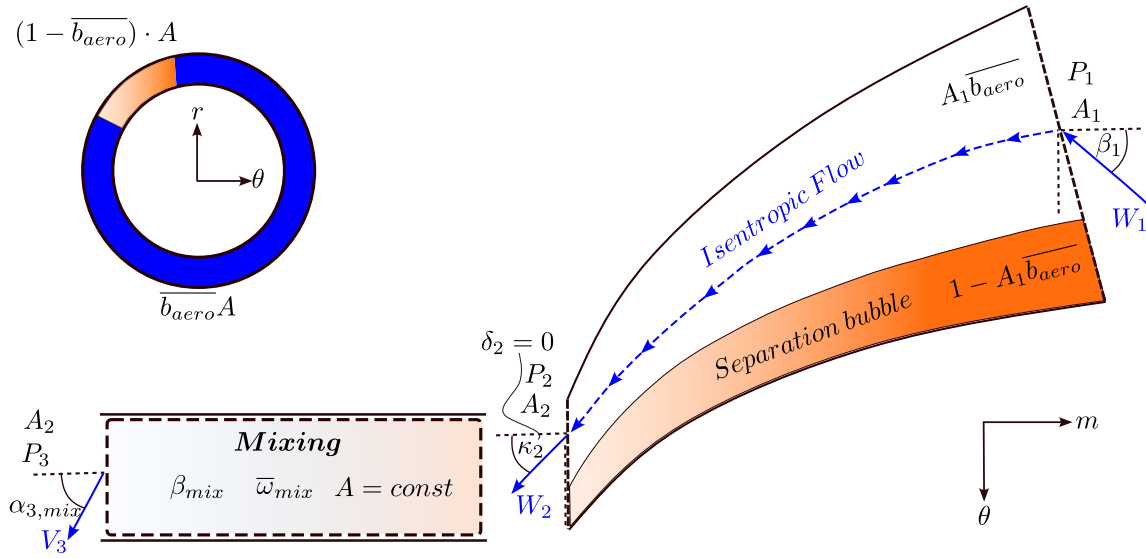


Figure 3.14: Schematic overview of the simplified axial blade-duct configuration showing the assumptions of the blockage-mixing model. Mixing occurs in an entirely axial, constant area duct downstream the blade LE.

Unlike Longley's [91] simplified model, the new model presented hereafter takes into account changes in blade angle, radius and area at the inlet and the outlet of the blade passage, making it universally applicable to any type of blade. A control volume analysis is performed on a notional blade passage. The control volume is confined within the geometrical boundaries of two adjacent blades as well as the inlet and outlet of the blade passage. The inlet flow-field properties (TE at reverse flow) are obtained just upstream of the geometrical inlet, where no aerodynamic blockage is present. Since a control volume approach is used, the conservation form of those equations is utilized to express the blockage parameter purely as a function of inlet and outlet geometric parameters and inlet flow-field variables. The blockage model is derived by combining conservation of energy, conservation of rotary stagnation pressure in the isentropic stream and conservation of momentum parallel to the streamwise coordinate along the full blade passage. In the equations presented hereafter, subscript 1 refers to the flow inlet at reverse flow conditions i.e. TE, subscript 2 refers to the flow outlet

at reverse flow conditions i.e. LE, while subscript 3 denotes the flow outlet of the control volume, where the flow exits the control volume completely mixed-out. Mass flow and momentum conservation in the streamwise direction in the blade passage are expressed by eqns.(3.133) and (3.134) respectively:

Mass flow conservation:

$$\rho W_1 \cos \beta_1 A_1 = \rho W_2 \cos \kappa_2 A_2 \overline{b_{sep}} \quad (3.133)$$

Momentum conservation along the streamwise direction:

$$P_2 A_2 \cos \kappa_2 - P_1 A_1 \cos \kappa_1 + \rho W_2^2 A_2 \overline{b_{sep}} \cos \kappa_2 - \rho W_1^2 \cos (\beta_1 - \kappa_1) \cos \beta_1 A_1 = \iiint \rho f_{cyl} \hat{l} dV \quad (3.134)$$

The volume integral source term in eqn.(3.134) is computed by eqn.(3.135) whereby the relative flow-path was assumed to coincide with that of the mean camberline (no deviation):

$$\iiint \rho f_{cyl} \hat{l} dV = \iiint \frac{\rho}{r} \sin \phi \left[\cos \kappa (w_\theta + \Omega r)^2 - V_m \sin \kappa (w_\theta + 2\Omega r) \right] \hat{l} dV \quad (3.135)$$

Conservation of rotary stagnation pressure in the isentropic bulk-flow:

$$P_1 + \frac{1}{2} \rho (W_1^2 - U_1^2) = P_2 + \frac{1}{2} \rho (W_2^2 - U_2^2) \quad (3.136)$$

Combining eqns.(3.133) - (3.136), the unknown flow-field variables at the outlet are eliminated and the aerodynamic blockage parameter, $\overline{b_{sep}}$, is estimated by solving the following quadratic function:

$$\begin{aligned} \overline{b_{sep}}^2 \left\{ \left(\frac{A_2 \cos \kappa_2}{A_1 \cos \beta_1} \right)^2 \left[2 \frac{P_1}{\rho W_1^2} \left(\frac{A_1 \cos \kappa_1}{A_2 \cos \kappa_2} - 1 \right) - \frac{(U_2^2 - U_1^2)}{W_1^2} + 2 \left(\frac{A_1 \cos \beta_1 \cos (\beta_1 - \kappa_1)}{A_2 \cos \kappa_2} \right) \right. \right. \\ \left. \left. + \frac{2}{A_2 \cos \kappa_2 W_1^2} \iiint f_{cyl} \hat{l} dV - 1 \right] \right\} - 2 \overline{b_{sep}} + 1 = 0 \end{aligned} \quad (3.137)$$

The solution of the quadratic function can be directly computed by:

$$\overline{b_{sep}} = \frac{1 \pm \sqrt{1 - \alpha}}{\alpha} \quad (3.138)$$

,where the coefficient α is given by eqn.(3.139)

$$\alpha = \overline{b_{sep}}^{-2} \left\{ \left(\frac{A_2 \cos \kappa_2}{A_1 \cos \beta_1} \right)^2 \left[2 \frac{P_1}{\rho W_1^2} \left(\frac{A_1 \cos \kappa_1}{A_2 \cos \kappa_2} - 1 \right) - \frac{(U_2^2 - U_1^2)}{W_1^2} + 2 \left(\frac{A_1 \cos \beta_1 \cos(\beta_1 - \kappa_1)}{A_2 \cos \kappa_2} \right) + \frac{2}{A_2 \cos \kappa_2 W_1^2} \iiint f_{cyl} \hat{l} dV - 1 \right] \right\} \quad (3.139)$$

The volume integral in eqn.(3.135) can either be approximated by direct integration, using previous time-step flow-field data or analytically, by setting the aerodynamic blockage parameter to 1 ($\overline{b_{sep}} = 1$) for a zero incidence angle. This ensures that that no aerodynamic blockage will be imposed when the incidence angle of the flow at the TE is zero. The estimated aerodynamic blockage is then multiplied by a mixing parameter to account for partial mixing effects as the flow moves along the blade passage. The imposed aerodynamic blockage is typically in the range $[0.3 \sim 0.5]$ of the estimated one.

$$\iiint f_{cyl} \hat{l} dV = \frac{\cos \kappa_2 A_2 W_1^2}{2} \left\{ \left(\frac{\cos \kappa_1 A_1}{\cos \kappa_2 A_2} - 1 \right)^2 - 2 \frac{P_1}{\rho W_1^2} \left(\frac{A_1 \cos \kappa_1}{A_2 \cos \kappa_2} - 1 \right) + \frac{U_2^2 - U_1^2}{W_1^2} \right\} \quad (3.140)$$

3.5.2 Deviation model

The mixing of the jet and separated stream precipitates changes in the circumferential and axial momentum and consequently in the relative flow angle, once the flow exits the blade passage and mixes-out. The mixed-out relative flow angle is estimated by performing a control volume analysis downstream the LE. It is reasonable to assume that a constant area mixing process takes place in the duct downstream the LE (reverse flow) of impellers and axial compressor blades due to the negligible change in the radius and area of the duct. The exact definition of the leftmost boundary of the control volume, corresponding to the flow outlet is redundant, since constant area mixing is assumed. The exact definition of those boundaries is however not trivial, when the constant-area mixing assumption is not valid. When estimating

the mixed-out angle of the flow exiting a vaned diffuser blade, the change in area is substantial (figure(3.15)). The placement of the outflow boundary dictates the change between the inlet and the outlet areas and therefore the consequential change in the relative flow angle due to the mixing process.

Two models are derived by applying continuity and circumferential momentum conservation at the inflow and outflow boundaries of the control volume defined by the two blades, the LE and either an arbitrarily positioned outflow boundary (constant area mixing) or a boundary defined by its radial position (radial, variable area mixing). The first assumes a strictly axial duct while the latter a strictly radial one. The radial configuration, representative of the blockage-mixing process in vaned diffusers, is illustrated in figure(3.15). The inflow boundary of the control volume (LE at reverse flow) is denoted with the subscript 2 and the outflow boundary with the subscript 3.

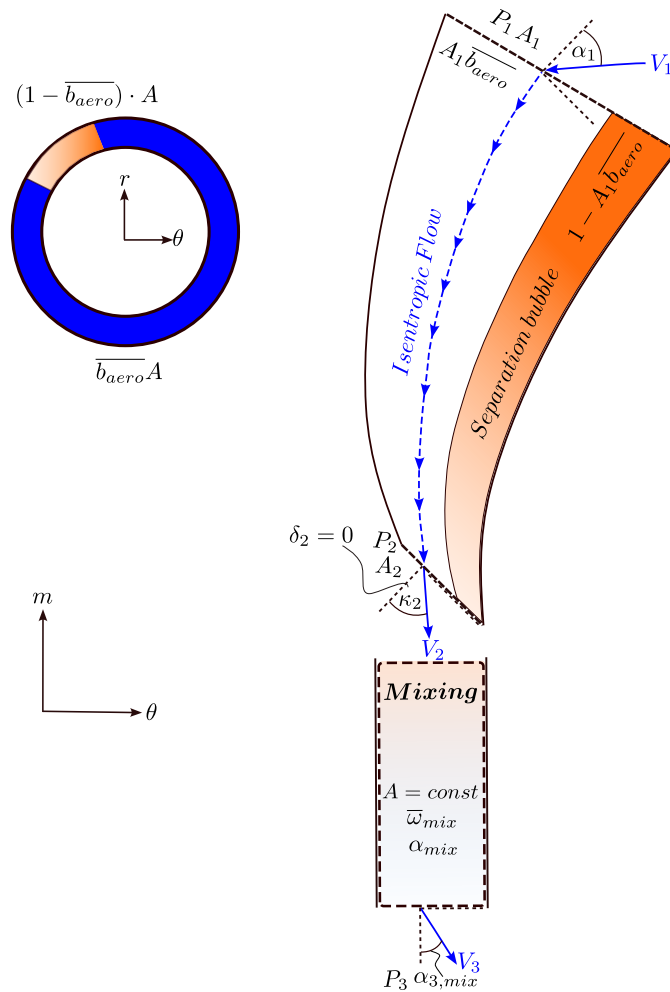


Figure 3.15: Schematic overview of the simplified radial blade-duct configuration showing the assumptions of the blockage-mixing model.

The conservation form of the mass flow and circumferential momentum equations are used to express the mixed-out flow angle as a function of geometric properties and the aerodynamic blockage parameter, $\overline{b_{sep}}$. Conservation of mass flow and circumferential momentum in a generic control volume located downstream the LE (reverse flow) and assuming separated flow (aerodynamic blockage) at the LE gives:

Mass flow conservation:

$$\rho V_3 \cos \alpha_{3,mix} A_3 = \rho V_2 \cos \kappa_2 \overline{b_{sep}} A_2 \quad (3.141)$$

Circumferential momentum conservation:

$$\rho V_{3m} V_{3\theta} A_3 = \rho V_{2m} V_{2\theta} A_2 \overline{b_{sep}} - \iiint \frac{\sin \phi}{r} V_m V_\theta dV \quad (3.142)$$

In the above equations, the angular velocity is zero ($\Omega = 0$) since the mixing is assumed to occur inside stationary, ducted components. Along a fully axial, constant-area duct, the inflow and outflow areas are the same. Since the duct is considered strictly axial, the flow path angle, ϕ , and consequently the volume integral term is zero. The second model assumes a strictly radial geometry, i.e. $\phi = 90^\circ$, whereby the areas of the control volume are purely a function of the radius. The substantial change in the areas of the inflow and outflow boundaries of the control volume do not allow the implementation the aforementioned constant-area mixing simplifications. In both cases, the flow is assumed to be perfectly guided by the blades when it exits the blade passage. The deviation angle for axial and radial flow mixing is estimated by eqns.(3.143) - (3.147). It should be noted, that the flow angles are absolute flow angles, since both the vaned diffuser and the vaneless space are not rotating.

Constant-area mixing:

$$\tan \beta_{3,mix} = \frac{\tan \kappa_2}{\overline{b_{sep}}} \quad (3.143)$$

$$\delta_{3,mix} = \text{atan} \left(\frac{\tan \kappa_2}{\overline{b_{sep}}} \right) - \kappa_2 \quad (3.144)$$

Radial, variable-area mixing:

$$\tan\alpha_{3,mix} = \tan\kappa_2 \frac{r_3}{r_2} \left(\frac{1}{\overline{b_{sep}}} - \frac{4}{\left(1 + \frac{r_3}{r_2}\right)^2} \left(1 - \frac{r_3}{r_2}\right) \right) \quad (3.145)$$

Due to the change in radius, the flow angle changes regardless of the presence or not, of mixing phenomena. The ideal flow angle at the outflow boundary can be simply computed by setting the blockage parameter in eqn.(3.145) equal to 1.

$$\tan\alpha_{3,id} = \tan\kappa_2 \frac{r_3}{r_2} \left(1 - \frac{4}{\left(1 + \frac{r_3}{r_2}\right)^2} \left(1 - \frac{r_3}{r_2}\right) \right) \quad (3.146)$$

The deviation angle due to the mixing process is the difference between the mixed-out flow angle and the ideal one:

$$\delta_3 = \alpha_{3,mix} - \alpha_{3,id} \quad (3.147)$$

3.5.3 Mixing loss model

During flow reversal, losses due to mixing are considered dominant, while other loss mechanisms such as boundary layer development are considered subsidiary. The reverse flow modelling approach, introduces losses, representative of the losses generated as a result of all the complex viscous-related flow phenomena during flow reversal via the turning force model. The additional losses due to mixing-out of the flow in the ducted region located just downstream the LE (reverse flow) are approximated following a similar approach presented in section (3.5.2). Performing a similar control volume analysis, assuming incompressible flow and no deviation at LE, the conservation form of continuity, meridional momentum and energy are combined to derive stagnation pressure loss coefficients due to mixing of the stagnant, separated flow and the isentropic, shear-free jet-flow in both purely axial and radial ducts. The constant-area and radial (variable area) loss coefficients due to mixing are computed by eqns.(3.148)-(3.149)

Constant-area mixing loss coefficient:

$$\overline{\omega}_{2-3} = \cos\kappa_2^2 \left[\overline{b_{sep}}^2 \left(2 - \frac{1}{\cos\beta_{3,mix}^2} \right) - 2\overline{b_{sep}} + \frac{1}{\cos\kappa_2^2} \right] \quad (3.148)$$

Radial, variable-area mixing loss coefficient:

$$\bar{\omega}_{2-3} = \frac{P_2 \left(1 - \frac{r_2}{r_3}\right)}{P_{02} - P_2} + \frac{r_2}{r_3} \cos \kappa_2^2 \left\{ \bar{b}_{sep}^2 \left[\frac{r_3}{r_2} \left(2 - \frac{1}{\cos \alpha_{3,mix}^2}\right) - 8 \tan \kappa_2^2 \frac{\left(1 - \frac{r_3}{r_2}\right)}{\left(1 + \frac{r_3}{r_2}\right)^2} \right] - 2\bar{b}_{sep} + \frac{r_3}{r_2} \frac{1}{\cos \kappa_2^2} \right\} \quad (3.149)$$

, where the definition of the pressure loss coefficient is performed using the compressible dynamic head as follows:

$$\bar{\omega}_{2-3} = \frac{P_{03} - P_{02}}{P_{02} - P_2} \quad (3.150)$$

The pressure loss coefficients predicted by eqns.(3.148) and (3.150) are a function of both $\bar{b}_{sep}$ and $\beta_{3,mix}$, $\alpha_{3,mix}$ and reduce to zero as $\bar{b}_{sep} \rightarrow 1$ i.e. when aerodynamic blockage approaches zero. The radius ratio accounts for the change in the areas of the inflow and outflow boundaries of the control volume; as $r_3/r_2 \rightarrow 1$, the contribution of the additional terms of eqn.(3.149) becomes inconsequential and for the limiting case, where $r_3/r_2 = 1$, eqn.(3.149) reduces to eqn.(3.148). The dynamic head in the denominator of the pressure loss coefficient has been replaced with the difference between the stagnation and the static pressure. Alternatively, the Prandtl-Glauert or Karman-Tsien compressibility corrections could be applied using either inflow, outflow or averaged flow properties. Corrections using outflow properties in compressor cascades were shown to yield more accurate results [114] although inflow flow-field properties could be utilized to avoid dependency of the model on outflow conditions. The use of eqns.(3.143) - (3.145) is preferred, since supersonic Mach numbers in the relative FoR are frequently encountered at reverse flow conditions. The alternative form of the mixing loss coefficients is given by:

Constant-area mixing loss coefficient:

$$\bar{\omega}_{2-3} = \cos \kappa_2^2 \left[\bar{b}_{sep}^2 \left(2 - \frac{1}{\cos \beta_{3,mix}^2}\right) - 2\bar{b}_{sep} + \frac{1}{\cos \kappa_2^2} \right] \quad (3.151)$$

Radial, variable-area mixing loss coefficient:

$$\bar{\omega}_{2-3} = \frac{P_2 \left(1 - \frac{r_2}{r_3}\right)}{\frac{1}{2} \rho V_2^2} + \frac{r_2}{r_3} \cos \kappa_2^2 \left\{ \bar{b}_{sep}^2 \left[\frac{r_3}{r_2} \left(2 - \frac{1}{\cos \beta_{3,mix}^2}\right) - 8 \tan \kappa_2^2 \frac{\left(1 - \frac{r_3}{r_2}\right)}{\left(1 + \frac{r_3}{r_2}\right)^2} \right] - 2\bar{b}_{sep} + \frac{r_3}{r_2} \frac{1}{\cos \kappa_2^2} \right\} \quad (3.152)$$

In eqns.(3.151) and (3.152) the incompressible pressure loss coefficient is defined as:

$$\bar{\omega}_{2-3,ic} = \frac{P_{03} - P_{02}}{\frac{1}{2}\rho V_2^2} \quad (3.153)$$

The loss coefficients using the Prandtl-Glauert or Karman-Tsien compressibility corrections and inflow flow-field properties are given by eqns.(3.154) and (3.155) respectively:

$$\bar{\omega}_{2-3} = \frac{\bar{\omega}_{2-3,ic}}{\sqrt{1 - Ma_{rel,in}^2}} \quad (3.154)$$

$$\bar{\omega}_{2-3} = \frac{\bar{\omega}_{2-3,ic}}{\sqrt{1 - Ma_{rel,in}^2} + \frac{Ma_{rel,in}^2}{\sqrt{1 - Ma_{rel,in}^2}} \frac{\bar{\omega}_{2-3,ic}}{2}} \quad (3.155)$$

3.6 Empirical correlations

The body force model presented in section (3.4) requires the estimation of essential compressor performance-related quantities ,i.e. aerodynamic blockage, deviation angle and stagnation pressure loss which are not a priori known. At forward flow conditions, these quantities are obtained by a set of component-specific empirical correlations, while a simplified aerodynamic blockage model provides estimates for the aerodynamic blockage inside impeller and vaned diffuser passages. Aerodynamic blockage in axial compressor passages is considered negligible, thus reduction in the effective area in axial compressor passages is considered only due to blade metal blockage. Dedicated loss and flow slippage/deviation models compatible with different compressor blade types e.g. axial rotors/stators, IGVs and OGVs, impeller blades and vaned diffuser blades are embedded into the code, providing estimations of the aforementioned quantities in a transient manner.

3.6.1 Pressure loss coefficients and deviation angle

Estimations of the stagnation pressure loss in blade passages due to various entropy generation mechanisms are obtained via dedicated empirical loss models applicable to the type of blades under consideration. Aungier's correlations [64, 68] were preferred over other models presented in the literature review, mainly due to their ability of providing accurate global aerodynamic performance predictions in 1D mean-streamline analysis but also due to their direct compatibility with the viscous force model

(eqn.(3.119)). Augnier's models express losses in the form of stagnation pressure loss coefficients in the relative FoR rather than stagnation enthalpy changes and are thus directly compatible with the viscous force model. Flow deviation in impeller blades is computed by transforming the slip factor into an equivalent deviation angle. Stations denoted with 1 refer to inlet (LE) conditions, while subscript 2 refers to outlet conditions. A global loss coefficient and deviation angle or slip factor is then computed for each blade passage and the spatial redistribution of the pressure loss and the deviation along individual cells is performed via distribution coefficients (see section(3.4.4)).

Axial rotors/stators and vaned diffusers

The loss coefficient in the relative FoR is defined as the ratio of the pressure drop (inlet to outlet), relative to the ideal one, over the compressible dynamic head at the inlet (eqn.(3.156)).

$$\bar{\omega}_{rel} = \frac{P_{02,rel} - P_{01,rel}}{P_{01,rel} - P_1} \quad (3.156)$$

In stator blades, the pressure loss coefficient is expressed using absolute FoR stagnation quantities. The cumulative stagnation pressure loss due to different loss sources is given by eqn.(3.157)).

$$\overline{\Delta P_{0,rel}} = (P_{01,rel} - P_1) \sum_i \bar{\omega}_{i,rel} \quad (3.157)$$

The deviation angle is directly obtained by correlations, with the imposed local relative flow angle being computed as per by eqn.(3.158).

$$\beta_{imp}(m, n) = \kappa(m, n) + C_{dev}^*(m, n) \bar{\delta} \quad (3.158)$$

Impellers

In impeller blades, the loss coefficient is defined in the relative FoR and is identical to the axial one, defined by eqn.(3.156). However, the cumulative stagnation pressure loss in the relative FoR needs to be adjusted by a correction factor to preserve the entropy rise unchanged [68] due to the change in the mean streamline radius. The relative stagnation pressure loss in impeller blades and the correction factor proposed by Augnier are given by eqn.(3.159) and eqn.(3.160) respectively.

$$\overline{\Delta P_{0,rel}} = f_c (P_{01,rel} - P_1) \sum_i \bar{\omega}_{i,rel} \quad (3.159)$$

The correction coefficient, f_c , and the pressure drop, $\Delta P_{0,rel}$ are defined as follows:

$$f_c = \frac{\rho_{02,rel} T_{02,rel}}{\rho_{01,rel} T_{01,rel}} \quad (3.160)$$

$$\overline{\Delta P_{0,rel}} = P_{02,rel}|_{id} - P_{02,rel}|_{act} \quad (3.161)$$

To define the imposed relative flow path, the deviation angle needs first to be determined. Flow misalignment in impeller blades is an inviscid phenomenon [115] and is expressed using the slip factor, σ (eqn.(3.162)). Flow slippage results in reduced flow turning and in turn, energy input into the flow due to reduction of the absolute circumferential velocity component at the outlet. Assuming that the meridional velocity is unaffected due to flow slippage, thus implying no changes in mass flow, the meridional velocity component available by the flow-field solution at every timestep, essentially represents the ideal, perfectly guided, meridional velocity component.

$$\sigma = \frac{u_{\theta 2}}{u_{\theta 2,id}} \quad (3.162)$$

To compute the relative flow angle at the given mass flow, the ideal swirl velocity components are computed using the solver's averaged meridional velocity component and a flow angle equal to the blade angle at the outlet, implying zero-slippage. The ideal circumferential component of the relative velocity vector is then computed as:

$$w_{\theta 2,id} = \tan(\kappa_2) V_{mer,2} \quad (3.163)$$

Introducing the slip factor, the actual relative velocity component (with respect to the correlations, not the one extracted from the solver) is computed as:

$$w_{\theta 2,act} = (w_{\theta 2,id} + \Omega r_2) \sigma - \Omega r_2 \quad (3.164)$$

The actual relative circumferential velocity component, $w_{\theta 2,act}$, in conjunction with $V_{mer,2}$ can be used to obtain the actual relative flow angle corresponding to the slip factor computed by the correlations. The deviation angle is determined by subtracting the actual flow angle, $\beta_{2,act}$ from the blade metal angle at the impeller's TE:

$$\bar{\delta} = \underbrace{\text{atan}\left(\frac{w_{\theta 2,act}}{V_{mer,2}}\right)}_{\beta_{2,act}} - \kappa_2 \quad (3.165)$$

3.6.2 Impeller correlations

Estimations of the stagnation pressure loss due to various entropy generation mechanisms inside impeller blades are acquired via a loss model accounting for 8 loss mechanisms; incidence loss, skin friction loss, clearance loss, blade loading loss, hub-to-shroud loading loss, mixing loss, choking and expansion loss. Shock wave losses are naturally introduced by the solver, thus the inclusion of a dedicated shock wave loss model is redundant. Velocity, density and flow angle properties are area-averaged while temperature and stagnation properties are mass-flow-averaged. The loss correlations embedded in the code and presented hereafter are primarily the ones developed and proposed by Aungier [68] but variations proposed by other authors are also adopted. Figure(3.16) depicts the nomenclature conforming with the correlations presented below. It is important to note, that the blade and flow angles in Aungier's correlations are expressed with respect to the circumferential axis and not with respect to the meridional axis.

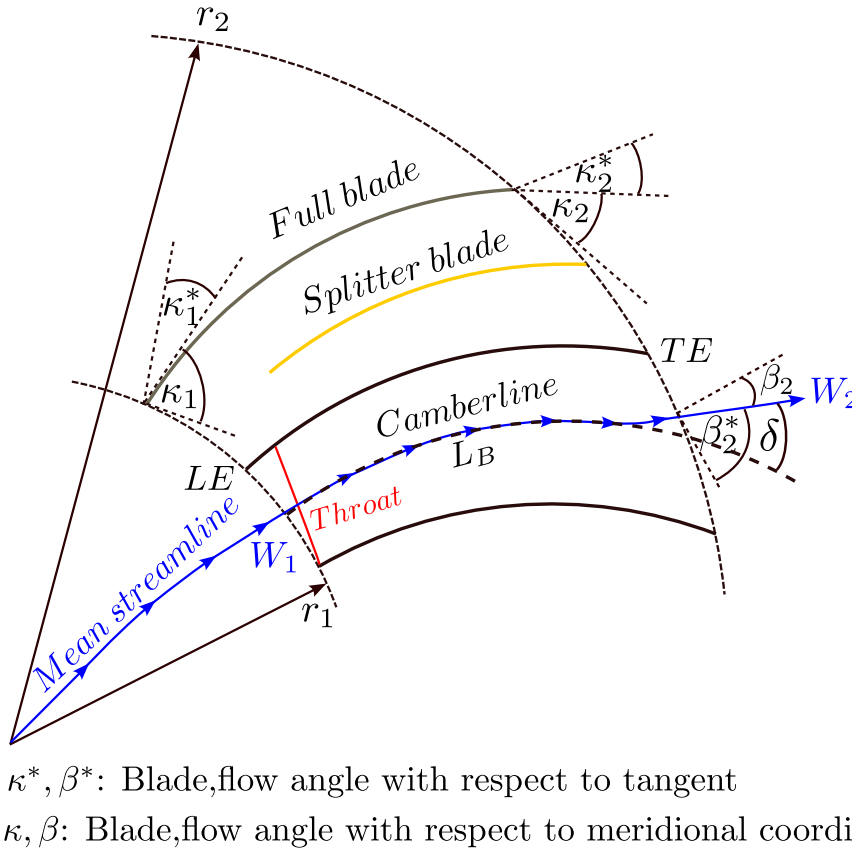


Figure 3.16: Definition of angles and impeller nomenclature in accordance to [68] .

Incidence Loss

The incidence loss accounts for flow adjustment losses due to mismatch of the inlet flow and the actual blade angle.

$$\bar{\omega}_{inc} = 0.8 \left[1 - \frac{V_{mer,1}}{W_1 \sin \beta_1} \right]^2 + \left[\frac{N_{FB} t_{b1}}{2\pi r_1 \sin \beta_1} \right]^2 \quad (3.166)$$

where, N_{FB} the number of full blades and t_{b1} the blade thickness at the inlet.

Skin friction Loss

The skin friction loss accounts for wall friction losses.

$$\bar{\omega}_{inc} = 4c_f \left(\frac{\bar{W}}{W_1} \right)^2 \frac{L_B}{d_{H,imp}} \quad (3.167)$$

where, c_f the skin friction coefficient, L_B the camber length of the full blades, $d_{H,imp}$ the impeller hydraulic diameter and $\bar{W}$ the average velocity along the passage. The skin friction coefficient is a function of the Reynolds number and is approximated by different equations when the flow is laminar, in the transition region or turbulent [68]. The turbulent skin friction correlation is taken from [116].

$$c_f = \begin{cases} \underbrace{\frac{16}{Re_d}}_{c_{fl}} & \text{if } Re_d \leq 2 \cdot 10^3 \\ c_f = c_{fl} + (c_{ft} - c_{fl}) \left(\frac{Re_d}{2000} - 1 \right) & \text{if } 2 \cdot 10^3 < Re_d \leq 4 \cdot 10^3 \\ \underbrace{0.0412 Re^{-0.1925}}_{c_{ft}} & \text{if } 4 \cdot 10^3 < Re_d \end{cases} \quad (3.168)$$

The hydraulic diameter is given by eqn.(3.169) as suggested by Zhang et al. [116]:

$$d_{H,imp} = \frac{D_2 \cos \beta_2}{\frac{N_{bl}}{\pi} + \frac{D_2 \cos \beta_2}{b_w}} + \frac{0.5 \left(\frac{D_{1t}}{D_2} + \frac{D_{1h}}{D_2} \right) \left(\frac{\cos \beta_{1t} + \cos \beta_{1h}}{2} \right)}{\frac{N_{bl}}{\pi} + \frac{D_{1t} + D_{1h}}{D_{1t} - D_{1h}} \frac{\cos \beta_{1t} + \cos \beta_{1h}}{2}} \quad (3.169)$$

,where N_{bl} the total number of blades and h, t subscripts denoting quantities at the hub and the tip respectively. The average velocity and density are estimated by:

$$\begin{cases} \bar{W}^2 = 0.5 (W_1^2 + W_2^2) \\ \bar{W}^2 \geq 0.5 (W_{th}^2 + W_2^2) \\ \bar{\rho}^2 = 0.5 (\rho_1^2 + \rho_2^2) \end{cases} \quad (3.170)$$

The Reynolds number (eqn.(3.171)) is computed using the averaged density and velocity while the hydraulic diameter replaces the characteristic length.

$$Re_d = \frac{\rho \bar{W} d_H}{\mu} \quad (3.171)$$

The air viscosity is computed using Sutherland's law. The coefficients are adjusted according to FLUENT theory guide [105]:

$$\begin{cases} \mu = \frac{C_1 \bar{T}^{3/2}}{\bar{T} + C_2} \\ \bar{T}^2 = 0.5 (T_1^2 + T_2^2) \\ C_1 = 1.458 \cdot 10^{-6} \\ C_2 = 110.4 \end{cases} \quad (3.172)$$

Clearance Loss

The clearance loss accounts for losses in open impellers due to clearance gap leakage.

$$\bar{\omega}_{CL} = \frac{2\dot{m}_{CL}\Delta P_{CL}}{\dot{m}\rho_1 W_1^2} \quad (3.173)$$

Where the velocity, pressure loss and mass flow of the clearance gap leakage flow are computed by the set of eqns.(3.174).

$$\begin{cases} u_{CL} = 0.816 \sqrt{2\Delta P_{CL}/\rho_2} \\ \Delta P_{CL} = \frac{\dot{m}(r_2 u_{\theta 2} - r_1 u_{\theta 1})}{N_{ef} \bar{r} \bar{b}_w L_m} \\ \bar{r} = (r_1 + r_2)/2 \\ \bar{b}_w = (b_{1w} + b_{2w})/2 \\ \dot{m}_{CL} = \rho_2 N_{ef} t_{CL} L_m u_{CL} \end{cases} \quad (3.174)$$

In the above equations, L_m is the blade mean streamline meridional length and N_{ef} the effective number of blades to account for the reduced size of the splitter blades. With the subscript sb , denoting splitter blade quantities, the effective number of blades is defined as follows:

$$N_{ef} = N_{fb} + N_{sb} \cdot L_{SB}/L \quad (3.175)$$

Impeller loading loss

The blade loading loss accounts for blade-to-blade pressure gradients which generate losses associated with secondary flows.

$$\bar{\omega}_{BL} = \frac{(\Delta W/W_1)^2}{24} \quad (3.176)$$

,where there average blade velocity difference, ΔW and blade input work coefficient I_B are evaluated using eqns.(3.177) and (3.178)

$$\Delta W = 2\pi D_2 u_{rot,2} I_B / (V_{ef} L_B) \quad (3.177)$$

$$I_B = \frac{u_{\theta 2}}{u_{rot,2}} - \frac{u_{\theta 1} u_{rot,1}}{u_{rot,2}^2} \quad (3.178)$$

Hub to shrouding loss

The hub-to-shroud loading loss is analogous to the blade loading loss model but in the hub to shroud direction.

$$\bar{\omega}_{HS} = \frac{1}{6} \left(\bar{\phi}_m \bar{bW} / W_1 \right)^2 \quad (3.179)$$

,where $\bar{\phi}_m$ is the mean streamline curvature, defined by eqn.(3.180)

$$\bar{\phi}_m = (\phi_2 - \phi_1)/L \quad (3.180)$$

Mixing loss

The mixing loss accounts for losses due to mixing of the free stream flow and blade wakes.

$$\bar{\omega}_{mix} = [V_{m,wake} - V_{m,mix}]^2 / W_1^2 \quad (3.181)$$

In eqn.(3.181), the meridional velocity of the wake, $V_{m,wake}$, is computed as function

of the separation velocity while the velocity of the mixed flow, $V_{m,mix}$ is computed from mass flow conservation as follows:

$$\left\{ \begin{array}{l} V_{m,wake} = \sqrt{W_{SEP}^2 - w_{\theta 2}^2} \\ V_{m,mix} = V_{m2} A_2 / (\pi D_2 b_{w2}) \\ W_{SEP} = W_2 \quad \text{if } D_{eq} \leq 2 \\ W_{SEP} = W_2 D_{eq} / 2 \quad \text{if } D_{eq} > 2 \\ D_{eq} = \frac{W_{max}}{W_2} \\ W_{max} = (W_1 + W_2 + \Delta W) \\ \Delta W = 2\pi D_2 u_{rot,2} I_B / (N_{ef} L_B) \end{array} \right. \quad (3.182)$$

,where D_{eq} is a generalized form of Lieblein's equivalent diffusion factor applicable to radial and mixed flow impellers and ΔW the average velocity difference between the pressure and the suction side.

Abrupt expansion loss

The mixing loss accounts for losses due to mixing of the distorted, separated flow at tip.

$$\bar{\omega}_\lambda = [(\lambda - 1) V_{m2} W_1]^2 \quad (3.183)$$

In the above equation, λ is the tip distortion factor and is associated with the tip clearance length, δ_{CL} and the tip area blockage, B_2 , which expresses aerodynamic blockage due to viscous effects with eqn.(3.185).

$$\lambda = 1 / (1 - B_2) \quad (3.184)$$

$$B_2 = \bar{\omega}_{sf} \frac{P V_1}{P V_2} \sqrt{\frac{W_1 d_H}{W_2 b_2}} + \left[0.3 + \frac{b_{w2}^2}{L_B^2} \right] \frac{A_R^2 \rho_2 b_{w2}}{\rho_1 L_B} + \frac{\delta_{CL}}{b_{w2}} \quad (3.185)$$

Choking loss

The choking loss accounts for losses as the Mach number at the throat approaches sonic conditions. The modified choking loss "trigger" proposed by Klausner and Gampe [117] is preferred over of the one proposed by Aungier, since it was found to

more accurately predict choking. The choking loss is approximated using Aungier's correlation in accordance to eqn.(3.186).

$$\bar{\omega}_{ch} = 0.5(0.005X + X^7) \quad (3.186)$$

The modified "choking trigger" variable, X , is defined as follows:

$$X = \frac{\dot{m}}{\dot{m}_{choke}} - \left(11 - 10 \frac{\dot{m}}{\dot{m}_{choke}}\right)^4 \quad (3.187)$$

The critical mass flow in the impeller is computed according to Dixon [16] by eqn.(3.188) where the $\overline{b_{aero}}$ term is added to account for reduction in the effective area due to the presence of aerodynamic blockage effects:

$$\dot{m}_{choke} = A_{th,imp} \overline{b_{aero}} N_{fb} \rho_{01} a_{01} \left[\frac{2 + (\gamma - 1)u_{rot}^2/a_{01}^2}{\gamma + 1} \right]^{(\gamma+1)/(2\gamma-2)} \quad (3.188)$$

In the above equation, a_{01} is the speed of sound computed using stagnation properties in the relative FoR and is computed according to eqn.(3.189):

$$a_0 = \sqrt{\frac{\gamma P_{0,rel}}{\rho_{0,rel}}} \quad (3.189)$$

Slip factor - Modified Wiesner's model

Wiesner's model constitutes one of the most widely used slip factor models in mean streamline analysis and is one of the two available models embedded in the code for flow slippage predictions. The slip factor remains rather invariable up to a limiting radius ratio, ϵ_{lim} , above which, it decreases abruptly. When the limiting radius ratio is exceeded, Aungier's alternative correlation is used.

$$\begin{cases} \sigma = 1 - \frac{\sqrt{\sin\beta_2 \sin\bar{\phi}_2}^{0.7}}{N_{bl}} & \frac{D_{in}}{D_{out}} \leq \epsilon_{lim} \\ \sigma_{cor} = \sigma \left[1 - \left(\frac{\epsilon - \epsilon_{lim}}{1 - \epsilon_{lim}} \right) \sqrt{\beta_2/10} \right] & \frac{D_{in}}{D_{out}} > \epsilon_{lim} \end{cases} \quad (3.190)$$

The limiting radius ratio, ϵ_{lim} , is given by eqn.(3.191):

$$\epsilon_{lim} = \frac{\sigma - \sigma^*}{1 - \sigma^*} \quad (3.191)$$

$$\sigma^* = \sin(19^\circ + 0.2\beta_2)$$

Slip factor - Unified slip factor model

The new unified model proposed by Qiu et al. [118] is applicable to axial, mixed and radial flow compressors and is the main slip factor model used in the code. It is preferred over other slip models because it considers the effect of the exit flow coefficient which according to Qiu et al., constitutes the primary driving element behind flow slippage variations at off-design conditions. It was validated against more than 90 different compressors with notable accuracy benefits over other models, primarily at off-design operation. The slip factor is computed as a function of three terms (eqn.(3.193)): the radial, the turning and the passage variation term, with the contribution of the latter being minuscule, thus it is omitted from eqn.(3.193) as per the authors' recommendation [118].

$$\sigma = 1 - \Delta\sigma_{radial} - \Delta\sigma_{turn} \quad (3.192)$$

The turning and the radial terms are given by eqn.(3.193):

$$\begin{cases} \Delta\sigma_{radial} = \frac{F\pi\cos\kappa_2\sin\phi_2}{N_{bl}} \\ \Delta\sigma_{turn} = \frac{Fs_2\Phi_2}{4\cos\kappa_2} \left(\frac{\partial\beta}{\partial m} \right)_2 \\ F = 1 - \sin\frac{\pi}{N_{bl}} \sin\left(\frac{\pi}{N_{bl}} + \kappa_2 \right) \cos\kappa_2 \sin\phi_2 - \frac{t_2}{s_2\cos\kappa_2} \end{cases} \quad (3.193)$$

,where F is the F -factor, s_2 the pitch at the outlet and Φ_2 the flow coefficient at the outlet.

Aerodynamic blockage

Impeller and vaned diffuser passages are subject to considerable reduction in the effective area due to aerodynamically-induced blockage. It typically is in the range 5% – 10%, i.e. $\overline{b_{aero}} = [0.9 - 0.95]$ [115], although experimentally measured aerodynamic blockage as high as 17% is reported for instance by Eisenlohr et al. [107] for the impeller SRV2. At forward flow conditions, aerodynamic blockage is modelled by a simplified contraction ratio correlation proposed by Aungier [68] which is a function of strictly geometric variables and consequently remains invariant regardless of the local flow-field variations.

$$\overline{b_{aero}} = \min \left\{ \sqrt{A_1 \sin \kappa_1 / A_{th,imp}} \quad , \quad 1 - (A_1 \sin \kappa_1 / A_{th,imp} - 1)^2 \right\} \quad (3.194)$$

3.6.3 Vaned diffuser correlations

Estimations of the stagnation pressure loss inside impeller blades are based on a loss model, which comprises the effect of: incidence loss, skin friction loss, mixing loss and choking loss. Shock wave losses are naturally introduced by the solver, thus the inclusion of such a model is omitted to prevent double-accounting. The same averaging process for inlet and outlet flow properties, discussed in section(3.6.2) is adopted. The loss correlations embedded in the code and presented hereafter are the ones developed and proposed by Aungier [68]. The subscript 3 designates flow/geometric properties at the inlet of the vaned diffuser, while the subscript 4 refers to properties at the vaned diffuser outlet.

Incidence loss

The incidence loss in vaned diffusers is analogous to the one in impellers and accounts for losses due to mismatch between the inlet flow angle and the blade angle at the LE. It is a function of the stall velocity V_S and the velocity at the optimum incidence, C_3^* and is estimated by eqn.(3.195):

$$\begin{cases} \overline{\omega}_{inc} = 0.8 [(V_3 - V_3^*) / V_3]^2 & \text{if } V_3 \leq V_{3S} \\ \overline{\omega}_{inc} = 0.8 \left[\left((V_3 / V_{3S})^2 - 1 \right) V_{th}^2 / V_3^2 + (V_{3S} - V_3^*)^2 / V_{3S}^2 \right]^2 & \text{if } V_{3S} < V_3 \end{cases} \quad (3.195)$$

The stall velocity is approximated by solving the system of the first four equations

shown in eqn.(3.196), while the minimum-loss flow angle is approximated by the last one.

$$\left\{ \begin{array}{l} K + K_0 = 0.39 \\ K = \frac{r_3}{h_{th}} \left[\frac{\cos\beta_3 S}{\cos\beta_{th}} - 1 \right] \\ K_0 = \frac{M_3^2 \sin^2 \kappa_3 \cos \kappa_3}{1 - M_3^2 \sin^2 \kappa_3} \\ \sin\beta_{th} = A_{th}/A_3 \\ \sin\beta_3^* = V_{m3}/V_3^* = \sqrt{\sin\beta_3 \sin\beta_{th}} \end{array} \right. \quad (3.196)$$

Skin friction loss

The skin friction loss model accounts for losses due to wall friction and is similar to the impeller model reviewed in the impeller loss section(3.6.2). A correction is applied here to account for the fact that unlike impeller passages, boundary layers profiles inside vaned diffuser passages are not fully developed. The correction is introduced via the inclusion of the $2\delta_{BL}/d_H$ term in eqn.(3.197)

$$\bar{\omega}_{SF} = 4c_f \left(\frac{\bar{V}}{V_3} \right)^2 L_B/d_H / (2\delta_{BL}/d_{H,dif})^{0.25} \quad (3.197)$$

,where δ_{BL} the boundary layer thickness, computed in the middle of the passage, based on a flat plate analysis. The correction term is approximated by eqn.(3.198):

$$2\delta_{BL}/d_H = \min \{ 5.142c_f L_b/d_{H,dif} \quad , \quad 1 \} \quad (3.198)$$

The hydraulic diameter is computed as the average value of the blade width at the throat and the outlet times two.

$$d_{H,dif} = b_{w,th} + b_{w,4} \quad (3.199)$$

The average velocity, $\bar{V}$, in the passage is estimated as an average value of the inlet and outlet velocities:

$$\bar{V} = \sqrt{0.5 (V_3^2 + V_4^2)} \quad (3.200)$$

Mixing loss

Wake mixing loss accounts for excessive streamwise diffusion and similar to the impeller model (3.6.2), only meridional components are assumed to contribute to the mixing process. It is a function of the separated and mixed-out flow velocity, $V_{m,mix}$.

$$\bar{\omega}_{mix} = [(V_{m,wake} - V_{m,mix})/V_3]^2 \quad (3.201)$$

In eqn.(3.201) the meridional velocity of the wake, $V_{m,wake}$ and the mixed-out velocity, $V_{m,mix}$ are a function of the separation velocity, V_{SEP} . These velocities can be computed by solving the system of equations shown in eqn.(3.202):

$$\left\{ \begin{array}{l} V_{m,wake} = \sqrt{V_{SEP}^2 - V_\theta^2} \\ V_{m,mix} = A_4 V_{m4} / (2\pi r_4 b_{w4}) \\ V_{SEP} = \max \{ V_3 / (1 + 2V_\theta) \quad , \quad V_{SEP} \geq V_4 \} \\ V_\theta = \{ 1, 2\theta_c / 11 \} \\ 2\theta_c = 2 \operatorname{atan} [((b_{w,b2b4} - t_{b4})b_{w4}/b_3 - b_{w,b2b3} + t_{b3}) / (2L_B)] \\ , \text{where } b_{w,b2b} = 2\pi r \sin \kappa / N_{ef} \end{array} \right. \quad (3.202)$$

Choking loss

The choking loss accounts for losses as the Mach number at the throat approaches sonic conditions and is identical to the impeller one, presented in section(3.6.2). The modified "choking trigger" of [117] is implemented, while the critical mass flow at the diffuser throat is determined by setting the angular velocity to zero, as suggested by Dixon [16]. The choking loss model and the "choking trigger" are identical, but the choking loss coefficient is included for the sake of completeness:

$$\bar{\omega}_{ch} = 0.5(0.005X + X^7) \quad (3.203)$$

The critical mass flow at the diffuser throat is estimated using Dixon's equation (eqn.(3.188)) by setting the angular velocity term to zero as follows:

$$\dot{m}_{choke} = A_{th,dif} \bar{b}_{aero} N_{fb} \rho_{01} a_{01} \left[\frac{2}{\gamma + 1} \right]^{(\gamma+1)/(2\gamma-2)} \quad (3.204)$$

Aerodynamic blockage

Aerodynamic blockage in vaned diffuser passages is approximated using the same contraction ratio correlation presented earlier:

$$C_r = \sqrt{A_3 \sin \kappa_3 / A_{th}} \quad (3.205)$$

Deviation

The deviation at the outlet of vaned diffuser passages is estimated using essentially axial flow compressor correlations modified accordingly to conform with the radial topology of the vaned diffuser blades. The flow angle at the outlet of the vaned diffuser blade is estimated by eqn.(3.206):

$$\beta_4 = \kappa_4 - \delta^* - \frac{\partial \delta}{\partial i} (\kappa_3 - \beta_3) \quad (3.206)$$

In eqn.(3.207), the minimum-loss deviation angle, δ^* , is computed using Howell's correlation:

$$\delta^* = \frac{\theta [0.92(a/c)^2 + 0.02(90^\circ - \kappa_4)]}{\sqrt{\sigma} - 0.02\theta} \quad (3.207)$$

The deviation angle gradient with respect to the incidence angle, $\frac{\partial \delta}{\partial i}$, location of the maximum camber, (a/c) , solidity, σ and the camber angle, θ , of a vaned diffuser blade are computed as follows:

$$\left\{ \begin{array}{l} \frac{\partial \delta}{\partial i} = \exp [((1.5 - \kappa_3/60)^2 - 3.3)\sigma] \\ a/c = [2 - (\bar{\kappa} - \kappa_3)/(\kappa_4 - \kappa_3)] / 3 \\ \sigma = N_{bl}(r_4 - r_3)/(2\pi r_3 \sin \bar{\kappa}) \\ \bar{\kappa} = 0.5(\kappa_3 + \kappa_4) \\ \theta = \kappa_4 - \kappa_3 \end{array} \right. \quad (3.208)$$

3.6.4 Inlet guide vane correlations

Dedicated inlet guide vane correlations proposed by Aungier [68] are incorporated in the loss correlation library of the code to estimate pressure loss and flow deviation in IGV passages. Axial compressor correlations are used but are generalized to broaden their applicability in axial, mixed and radial configurations. Unlike axial compressor correlations, blade and flow angles in impeller, vaned diffuser and inlet

guide vane correlations are expressed with respect to the tangent direction; therefore a blade correction angle is performed prior to the application of the axial compressor correlations as per eqn.(3.209). Note that the subscript -1 refers to the inlet and 0 to the outlet of the inlet guide vane:

$$\left\{ \begin{array}{l} K = \frac{k_0 - k_{-1}}{|k_0 - k_{-1}|} \\ \kappa_{-1} = K(90^\circ - \kappa_{-1}) \\ \kappa_0 = K(90^\circ - \kappa_0) \\ \bar{\kappa} = K(90^\circ - \bar{\kappa}) \\ \theta = \kappa_4 - \kappa_3 \end{array} \right. \quad (3.209)$$

The blade camber angle, θ , stagger angle, γ , solidity, σ_{sol} and position of the maximum camber line, a/c are determined by eqn.(3.210). Alternatively, the position of the maximum camber can be numerically estimated utilizing the 3D coordinates of the blade.

$$\left\{ \begin{array}{l} \theta = (\kappa_{-1} - k_0) \\ \gamma = (\kappa_{-1} + \bar{\kappa} + \kappa_0)/3 \\ \sigma_{sol} = \frac{N_{bl}(m_0 - m_{-1})}{2\pi r_0 \cos(\gamma)} \\ a/c = (2 - |\bar{\kappa} - \kappa_0|/\theta)/3 \end{array} \right. \quad (3.210)$$

Pressure loss coefficient

A single total pressure loss coefficient, representative of all the entropy generation mechanisms manifesting along the passage is derived, using axial compressor-based correlations as proposed by Aungier. The total pressure loss coefficient of IGV blades is computed by eqn.(3.216).

$$\bar{\omega} = \frac{0.018\sigma(1 + X^3)}{\cos(\kappa_0 + \delta^*)} \left[\frac{A_{-1}\cos(\kappa_{-1} + i^*)}{A_0\cos(k_0 + \delta^*)} \right]^2 D_{eq}^2 \quad (3.211)$$

In the above equation, X coefficient is given by eqn.(3.212)

$$\begin{cases} X = (i - i^*)/(i_S - i^*) & \text{if } i \geq i^* \\ X = (i^* - i)/(i^* - i_C) & \text{if } i < i^* \end{cases} \quad (3.212)$$

and the minim loss, i^* , positive stall, i_{PS} , negative stall, i_{NS} and actual incidence angle, i , are given by eqn.(3.213):

$$\begin{cases} i^* = (3.6 + \theta/3.42)\sqrt{\sigma} + \gamma - \kappa_{-1} \\ i_{PS} - i^* = 4(a/c)^2(10^\circ + 0.36\theta) \\ i^* - i_{NS} = 2(a/c)(9^\circ - 0.06\theta) \\ i = K(90^\circ - \beta_{-1}) - \kappa_{-1} \end{cases} \quad (3.213)$$

The equivalent diffusion factor of Lieblein is used and is defined as follows:

$$D_{eq} = \frac{[1.12 + 0.61\Gamma \cos^2(k_{-1} + i)] A_0 \cos(k_0 + \delta^*)}{A_{-1} \cos(\kappa_{-1} + i)} \quad (3.214)$$

where $\Gamma = [\tan(\kappa_{-1} + i) - r_0 A_0 \tan(\kappa_0 + \delta^*) / (r_{-1} A_{-1})] / \sigma$

The outlet flow angle at the TE of the passage is computed using Howell's correlation for axial compressor blades:

$$\beta_0 = 90^\circ - K(\kappa_0 + \delta^*) \quad (3.215)$$

,where the deviation angle at design conditions, δ^* is given by the following equation:

$$\delta^* = \frac{\theta [0.92(a/c)^2 + 0.02\kappa_0]}{\sqrt{\sigma} - 0.02\theta} \quad (3.216)$$

3.6.5 Axial compressor rotor/stator correlations

To extend the aerodynamic performance prediction capabilities of the code to axial compressor rotor, stator and fan blades, estimations of the pressure losses and the flow deviation along individual blade rows are requisite. These quantities are transiently predicted via a set of loss and deviation correlations proposed by Aungier [64]. These are predominantly correlations and data of Lieblein and Johnsen and

Bullock, but are combined with modifications and additional correlations proposed by Aungier, derived by utilizing and curve-fitting available experimental data. Axial compressor correlations are based on extensive 2D compressor cascade experimental data, primarily of NACA-type aerofoils, consequently adaptation methods are required to accommodate the improved aerodynamic performance capabilities of modern, more sophisticated aerofoil designs [64]. Axial compressor correlations are expressed with respect to design conditions, where losses and flow deviation are expected to be minimum. Off-design losses and flow deviation are then estimated as a function of their corresponding design quantities and are subsequently corrected using coefficients with accordance to the aerofoil geometry and profile under consideration. Correction coefficients are also included in the empirical models, essentially constituting tuning coefficients which substantially alter pressure loss and deviation predictions and in turn, the aerodynamic performance predictions of the cascade. A schematic overview of an axial compressor 2D cascade along with the nomenclature adopted hereafter is shown in figure(3.17)

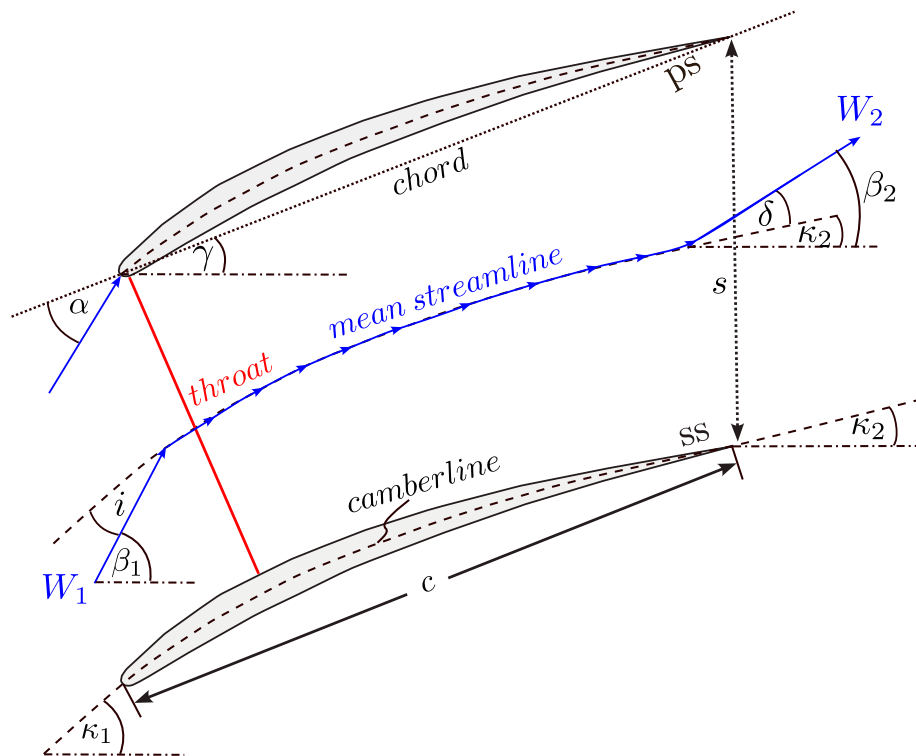


Figure 3.17: Definition of angles and blade nomenclature in accordance to [64] .

Pressure Loss

The embedded axial compressor loss model comprises a tip clearance loss and a profile loss model, with the latter representing the principal loss source. The profile loss is a function of primarily the blade geometry and flow-field variables during

off-design operation. The profile loss correlation is scaled with two coefficients K_1 and K_2 , which ideally should be calibrated according to local aerofoil geometrical data ,e.g. in different spanwise positions or possibly flow properties e.g. incidence angle. Aungier proposes two sets of K_1 and K_2 according to whom, yield rather accurate estimations for a range of different types of aerofoils. The first is a set of fixed values given by eqn.(3.217)

$$K_{set,1} = \begin{cases} K1 = 0.004 \\ K2 = 4 \end{cases} \quad (3.217)$$

The second set of coefficients consists of a constant K_1 coefficient and a K_2 coefficient as a function of certain blade geometric parameters:

$$K_{set,2} = \begin{cases} K1 = 0.0073 \\ K2 = 1 + (s/h)\cos\beta_2 \end{cases} \quad (3.218)$$

The diffusion factor at design point conditions, D^* , is computed according to Lieblein's definition using design condition velocities (the superscript $(*)$ indicates design conditions):

$$D^* = 1 - \frac{W_2^*}{W_1^*} + \frac{w_{\theta_1}^* - w_{\theta_2}^*}{2\sigma_{sol}W_1^*} \quad (3.219)$$

The design profile loss coefficient is subsequently estimated by eqn.(3.220):

$$\omega^* = [K_1(K_2 + 3.5D^* + 37D^{*4})] \frac{2\sigma}{\cos(\beta_2^*)} \quad (3.220)$$

When the incoming incidence is within a range which does not differ substantially from the design incidence angle, i^* , the variations in the loss coefficient are moderate. Once the incoming incidence angles exceed that range, the loss coefficient increases substantially. The numerical boundaries of the low-loss region are determined via a positive stall, i_{PS} and negative stall i_{NS} incidence angle respectively. The corresponding angle of attacks, α_{PS} and α_{NS} are estimated from eqns.(3.221)-(3.222):

$$\alpha_{NS} - \alpha^* = -9 + \left[1 + \left(\frac{30}{\beta_{1,NS}} \right)^{0.48} \right] \cdot \frac{\theta}{4.176} \quad (3.221)$$

$$\alpha_{PS} - \alpha^* = 10.3 + \left(2.92 - \frac{\beta_{1,S}}{15.6} \right) \cdot \frac{\theta}{8.2} \quad (3.222)$$

The above equations are solved iteratively, since the incoming relative flow angle is a function of both the incidence angle and the angle of attack $\beta_1 = i + \gamma$. The design incidence angle is computed by eqn.(3.223) as a function of the inlet flow angle, blade geometric parameters and a base, zero-camber aerofoil incidence angle at design conditions, $(i_0^*)_{10}$:

$$i^* = K_{sh} K_{t,i} (i_0^*)_{10} + n\theta \quad (3.223)$$

The base, zero-camber aerofoil incidence angle at design conditions is approximated by:

$$(i_0)_{10} = \frac{\beta_1^p}{5 + 46 \exp(-2.3\sigma_{sol})} - 0.1\sigma^3 \exp[(\beta_1 - 70)/4] \quad (3.224)$$

,where

$$n = 0.026\sigma_{sol} - 0.06 - \frac{(\beta_1/90)^{(1+1.2\sigma)}}{1.5 + 0.43\sigma_{sol}} \quad (3.225)$$

$$p = 0.914 + \sigma_{sol}^3/160$$

The positive and negative stall margins are given by:

$$R_{NS} = i^* - i_{NS} = i^* - i_{NS} \quad (3.226)$$

$$R_{PS} = i_{PS} - i^* = i_{PS} - i^* \quad (3.227)$$

These correlations are applicable to low Mach number cascades and compressibility corrections are required to extent their applicability to high-Mach number axial compressor cascades. The positive and negative stall incidence angles are readjusted using the following correction:

$$i_{NS} = \frac{i^* - R_{NS}}{1 + 0.5Ma_{1,rel}^3} \quad (3.228)$$

$$i_{PS} = i^* - \frac{R_{PS}}{1 + 0.5K_{sh}Ma_{1,rel}^3} \quad (3.229)$$

,where K_{sh} is a shape factor with a value of 1.0 for NACA profile, 1.1 for C4 series

and 0.7 for double circular arc profiles. The Mach number-corrected minimum loss incidence angle and pressure loss coefficient are obtained as follows:

$$i_m = i_{NS} + ((i_{PS} - i_{NS})R_{NS})/(R_{NS} + R_{PS}) \quad (3.230)$$

$$\omega_m = \omega^* \cdot \left(1 + \frac{(i_m - i^*)^2}{R_{PS}^2}\right) \quad (3.231)$$

Using the corrected minim loss coefficient, ω_m as a reference, the loss coefficient at different corrected optimum incidence angles is computed as a function of ω_m and the corrected minimum-loss incidence angle as follows:

$$\begin{cases} \bar{\omega} = \bar{\omega}_m[1 + \xi^2] & \text{if } -2 \leq \xi \leq 1 \\ \bar{\omega} = \bar{\omega}_m[5 - 4(\xi + 2)] & \text{if } \xi < -2 \\ \bar{\omega} = \bar{\omega}_m[2 + 2(\xi - 1)] & \text{if } \xi > 1 \end{cases} \quad (3.232)$$

,where ξ the normalized incidence angle defined as per eqn.(3.233)

$$\begin{cases} \xi = (i - i_m)/(i_{PS} - i_m); & \text{if } i \geq i_m \\ \xi = -(i - i_m)/(i_{NS} - i_m); & \text{if } i < i_m \end{cases} \quad (3.233)$$

Deviation

The deviation angle is estimated following a similar procedure, whereby the optimum-design deviation angle, δ^* is first estimated and depending on the operating conditions, the deviation angle is expressed as a function of δ^* . The design deviation angle is estimated with respect to a zero-camber aerofoil and adjustments in the form of correction factors are then applied. The base, zero-camber deviation angle is given by eqn.(3.234):

$$(\delta^*)_{1.0} = 0.01\sigma|\beta_1^*| + [0.74\sigma^{1.9} + 3\sigma] \cdot \left(\frac{|\beta_1^*|}{90}\right)^{(1.67+1.09\sigma)} \quad (3.234)$$

This is then adjusted via correction factors to extent its applicability to any aerofoil type. The design deviation angle is estimated as follows:

$$\delta^* = K_{SH}K_{t,\delta} \cdot (\delta^*)_{1.0} + m \cdot \theta \quad (3.235)$$

,where

$$b = 0.9625 - 0.17 \frac{|\beta_1^*|}{100} - 0.85 \left(\frac{|\beta_1^*|}{100} \right)^3 \quad (3.236)$$

$$m = \frac{0.17 - 0.0333 \frac{|\beta_1^*|}{100} + 0.333 \frac{|\beta_1^*|}{100^2}}{\sigma_{sol}^b} \quad (3.237)$$

In eqn.(3.235), K_{sh} is the shape coefficient discussed in the pressure loss section, while $K_{t,\delta}$ is a thickness correction factor computed as follows:

$$K_{t,\delta} = 6.25 \cdot \frac{t_b}{c} + 37.5 \cdot \left(\frac{t_b}{c} \right)^2 \quad (3.238)$$

The deviation at off-design conditions is subsequently computed as a function of the design deviation angle, δ^* , incidence angle i , gradient of the deviation angle with respect to the incidence angle at design conditions, $\left[\frac{\partial \delta}{\partial i} \right]^*$ and meridional velocity ratio, $\frac{V_{mer,2}}{V_{mer,1}}$ in accordance to eqn.(3.239):

$$\delta = \delta^* + \left[\frac{\partial \delta}{\partial i} \right]^* (i - i^*) + 10 \left(1 - \frac{V_{mer,2}}{V_{mer,1}} \right) \quad (3.239)$$

, where the variation of the deviation angle with incidence at design conditions is given by eqn.(3.240).

$$\left[\frac{\partial \delta}{\partial i} \right]^* = \frac{1 + (\sigma_{sol} + 0.25\sigma_{sol}^4) \left(\frac{|\beta_2^*|}{53} \right)^{2.5}}{e^{3.1\sigma_{sol}}} \quad (3.240)$$

3.7 Forward, mixed and reverse flow modelling

The global aerodynamic performance of a compressor is predominantly determined by three aspects; the energy input into the flow and thereby the flow turning inside rotors, the entropy generation which manifests in the form of stagnation pressure loss relative to the ideal, isentropic one and the effective annulus area which ultimately dictates the mass flow capacity of the compressor and is a function of both metal and aerodynamic blockage. Depending on the direction of the flow, at the LE and TE of the blade passage, the flow regime is classified as: forward, mixed inwards, mixed outwards and reverse (figure(3.18)). The classification of the flow direction

at the LE and TE boundaries is determined utilizing the velocities normal to the cell faces available from the solution of the local Riemann problem; these are the velocities perpendicular to the cell faces coinciding with the meridional geometric projection of the blade LE and TE (adaptive mesh). The characterisation of the blade passage regime is based on the flow direction of the weighted-averaged (mass flow or area averaged) velocity components (see next sections). A concurrent local flow direction evaluation along individual geometric streamlines, is also performed to ensure modelling consistency when the local flow regime is inconsistent with the global passage one. This prevents for example locally imposing a deviation angle at forward flow conditions if along a geometrical streamline e.g. centroids at tip, the regime of the flow is either mixed or reverse. The modelling of the flow regimes comprises a compound of empirical loss and deviation/slip correlations, analytical models and other methods which ultimately dictate the imposed relative path, the losses and the available area along the flow passage due to blade and aerodynamic blockage.

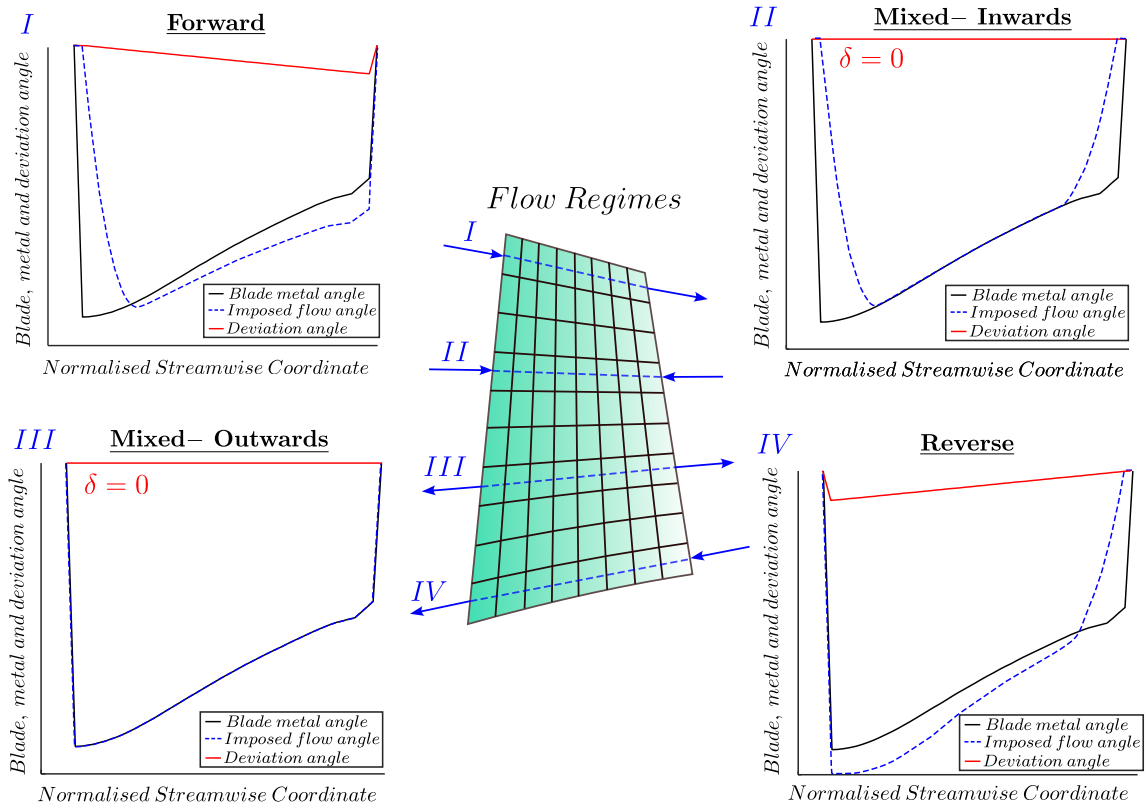


Figure 3.18: Imposed flow angle and deviation at different flow regimes.

3.7.1 Forward flow modelling

The global forward flow regime status inside a blade passage is assigned when the ratio of the forward over the total mass flow at the LE and TE exceeds a prescribed value. The aforementioned modelling aspects, i.e. deviation/slip, pressure loss and aerodynamic blockage are contingent upon flow-field data and are re-estimated at every timestep by the empirical correlations discussed in the previous section. To prevent numerical instabilities, a modified blade curvature region at the LE is imposed and is mathematically defined using a custom shape function (see section (3.7.4)).

It is common practice in axial compressor through-flow modelling codes to locally obtain deviation and pressure loss coefficients at different spanwise positions based on inlet and outlet flow-field and geometric properties. The inlet/outlet properties are obtained following either actual streamlines e.g. SLC codes [78] or geometric/grid streamlines [27]. The latter was deemed physically inaccurate and consequently incompetent particularly for impeller blades, due to the exacerbated mismatch of gridlines and actual streamlines. A single pressure loss coefficient and deviation/slip estimation is derived for every blade row using mass flow and area-averaged inlet and outlet flow-field properties. The locally imposed deviation and pressure loss is subsequently redistributed using either CFD-derived or custom distribution coefficients.

3.7.2 Reverse flow modelling

The global flow regime inside a blade passage is classified as reverse when the ratio of the reverse over the total mass flow at the LE and TE exceeds a prescribed value. Contrary to forward flow modelling, dedicated correlations for deviation, pressure loss and aerodynamic blockage at reverse flow conditions are not available in the literature for axial nor for impeller and vaned diffuser blades. Losses inside blade passages are introduced via the turning force, by projecting the turning force perpendicular to the blade instead of the flow. This misalignment generates flow separation and a component parallel to the relative flow-path acting as an entropy-generation force, representative of the cumulative effect of the various loss mechanisms present at reverse flow conditions. This modelling method is used for axial compressor rotating stall and surge modelling by Longley [91] and Righi [27]. Additional losses in the ducts adjoining bladed domains, accounting for additional mixing losses in downstream ducted components due to incomplete mixing-out of the jet flow and the separated flow are estimated using the analytical model presented in section(3.5.3). Flow deviation

due to mixing and aerodynamic blockage due to flow separation are estimated by the analytical model presented in section (3.5). The analytical blockage-mixing model is used strictly if the global passage regime is reverse while the deviation, additional mixing losses and aerodynamic blockage are updated in a transient manner. The radii terms in the deviation and pressure loss coefficient equations are set equal to the inflow and outflow radial position of the duct; for vaned diffuser blades, r_2 is the radius at the vaned diffuser LE and r_3 the radius at the impeller TE. The blockage coefficient is set in the range ($0.3 \sim 0.5$).

3.7.3 Mixed flow modelling

The global flow regime inside a blade passage is characterized as mixed, whenever simultaneous inflow or outflow occurs at the LE and the TE and the criteria neither of forward nor of reverse flow regime are met. The flow regime is further sub-categorised as mixed flow-inwards or mixed flow-outwards when the flow is simultaneously entering and exiting the blade passage respectively. Due to the unpredictable, chaotic nature during mixed flow, the modelling approach, proposed by Righi [27] is adopted. Flow deviation is considered negligible, while losses are purely introduced via the turning force model, by projecting the turning force normal to the blade. To ensure numerical stability, the curvature of the blade in the vicinity of both the LE and the TE is modified using the cubic shape function (mixed inwards only) given by eqn.(3.242). The aerodynamic blockage is kept constant and equal to the blockage imposed before the change of flow regime to mixed.

3.7.4 Shape functions

To prevent instantaneous excessive flow turning at the LE/TE (depending on the flow direction) which would inevitably lead to high turning forces and potentially numerical instabilities, the blade angle distribution along the streamwise coordinate is modified using shape functions. The shape functions generate a modified, artificial curvature, whereby the initial angle is that of the incoming flow just upstream the LE (forward flow) or just downstream the TE (reverse flow). The extent of the modified curvature region is controlled via a normalized control point, $\xi = [0 - 1]$, along the streamwise coordinate at each spanwise position, with 0 designating the LE and 1 the TE. Large transition regions enhance numerical stability due to the extended smoother modified curvature region, but may considerably affect the choking mass flow of the compressor. To minimize the impact of the modified curvature on the

compressor's local flow-field and global performance predictions, a higher-resolution discretization in terms of streamwise elements inside bladed regions is required to minimize the extent of the modified curvature region without compromising numerical stability ($\sim 80+$ streamwise elements in bladed domains). The modified curvature, expressed in terms of modified blade/imposed relative flow angle, is mathematically modelled by two polynomial functions; one quadratic and one cubic. The imposed relative flow angle essentially is expressed by a two-branch function whereby the imposed flow angle in the region delimited by the LE/TE and the transition point, ξ_{tr} , is analytically described by the shape function, while the imposed flow angle in the region delimited by the transition point and the TE/LE is numerically described by the actual imposed flow angle (eqn.(3.241)).

$$\beta_{imp} = \begin{cases} \beta_{imp}^* & \text{if } 0 \leq \xi < \xi_{tr} \\ \kappa(m, n) + C_{\delta}^*(m, n) \delta|_{TE} & \text{if } \xi_{tr} \leq \xi \leq 1 \end{cases} \quad (3.241)$$

The imposed relative flow angle, assuming zero deviation, during reverse flow at midspan of an impeller blade is shown in figure(3.19). Three modified curvature regions are shown, extending 10%, 30% and 60% of the blade's stream length from the TE.

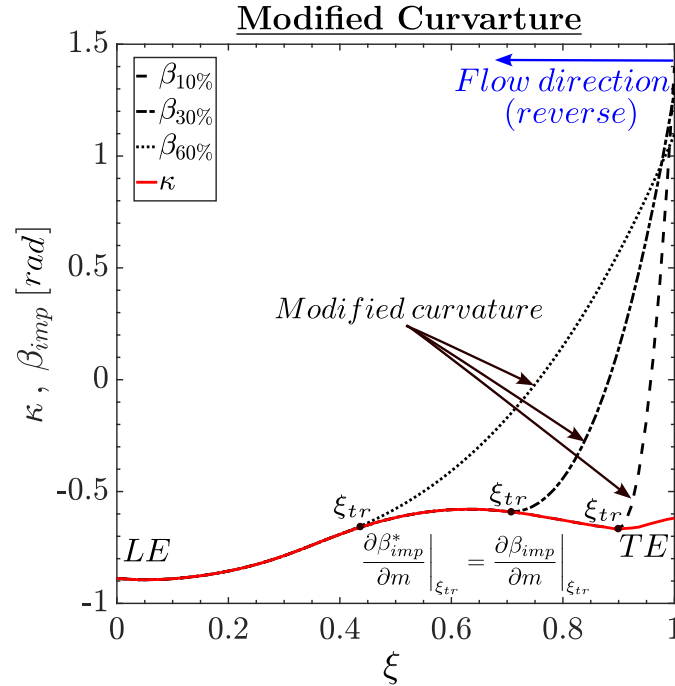


Figure 3.19: Modified curvature distributions extending 10% 30% and 60% in the streamwise direction from the TE. The flow is entering from the TE (reverse flow).

The definition of the modified blade curvature function requires three boundary conditions if the shape function is quadratic and four, if the shape function is cubic. The quadratic shape function can be fully defined by imposing the following boundary conditions: at the LE ($\xi = 0$) the blade angle is equal to the incoming relative flow angle, $(\beta|_{LE})$, at the transition point, ξ_{tr} , the flow angle is equal to the imposed one, $(\beta_{imp}|_{\xi_{tr}})$, while at the TE ($\xi = 1$) the flow angle is equal to the imposed, non-modified one, $(\beta_{imp}|_{\xi_{TE}})$. The points $\xi = (0, \xi_{tr}, 1)$ essentially constitute the control points of the function, with the transition point, ξ_{tr} , controlling the extent of the transition region. The same boundary conditions are used to define the cubic shape function along with an additional one; the gradient of the angle at the transition point with respect to the normalised streamwise coordinate is zero. This prevents potential gradient discontinuities at the point adjoining the modified and actual curvature regions, providing a significant advantage over the quadratic shape function. The cubic shape function is the default mathematical function embedded in the code which provides the modified distribution of the imposed relative flow angle. The modified imposed flow angle, β_{imp}^* is given by eqn.(3.242) along the streamwise direction at every geometrical, grid-defined, span position (hub to shroud) inside bladed domains. The meridional gradient of the imposed flow angle at the transition point is approximated using central finite differencing. The flow angle in the modified curvature region is obtained from eqn.(3.242).

$$\beta_{imp}^*(\xi) = c_1 + c_2\xi + c_3\xi^2 + c_4\xi^3 \quad (3.242)$$

,where ξ is the normalized blade meridional coordinate defined so that $\xi|_{LE} = 0$ and $\xi|_{TE} = 1$. The normalized blade meridional coordinate, ξ , is defined using the global meridional coordinate, m as follows:

$$\xi = \frac{m - m_{LE}}{m_{TE} - m_{LE}} \quad (3.243)$$

The boundary conditions for the definition of the cubic shape function summarised below:

$$\left\{ \begin{array}{l} \beta_{imp}^*|_{\xi=0} = \beta_{LE} \\ \beta_{imp}^*|_{\xi=\xi_{tr}} = \beta_{imp}|_{\xi_{tr}} \\ \frac{\partial \beta_{imp}^*}{\partial m} \Big|_{\xi_{tr}} = \frac{\partial \beta_{imp}}{\partial m} \Big|_{\xi_{tr}} \\ \beta_{imp}^*|_{\xi=1} = \beta_{TE} \end{array} \right. \quad (3.244)$$

Using the boundary conditions of eqn.(3.244) the coefficients c_1, c_2, c_3 and c_4 can be expressed in parametric form, as a function of the normalized transition point position, ξ_{tr} and the flow angles at the control points $\xi = (0, \xi_{tr}, 1)$, by solving a system of four equations (eqn.(3.245)). The modified imposed flow angle, β_{imp} , is recomputed at every timestep, to provide an updated imposed relative flow angle distribution since the incoming flow angle as well as the predicted deviation angle change at every timestep:

$$\left\{ \begin{array}{l} c_1 = \beta_{LE} \\ c_2 = \beta_{imp}|_{TE} - \beta_{LE} - c_3 - c_4 \\ c_3 = \frac{\frac{\partial \beta}{\partial m} \Big|_{\xi_{tr}} - \beta_{imp}|_{TE} + \beta_{LE} - c_4 (3\xi_{tr}^2 - 1)}{2\xi_{tr} - 1} \\ c_4 = \frac{(2\xi_{tr} - 1) \frac{(\beta_{imp}|_{\xi_{tr}} - \beta_{LE}) / \xi_{tr} - \beta_{TE} + \beta_{LE}}{\xi_{tr} - 1} - \frac{\partial \beta}{\partial m} \Big|_{\xi_{tr}} + \beta_{TE} - \beta_{LE}}{(\xi_{tr} + 1)(2\xi_{tr} - 1) - (3\xi_{tr}^2 - 1)} \end{array} \right. \quad (3.245)$$

3.8 Plenum and throttle model

Simulations of compressor post-stall aerodynamic behaviour requires modelling of a plenum and a throttle, in order to enable dynamic change of the imposed boundary conditions during the simulation of a surge event. The necessity for the inclusion

of that modelling aspect also stems from the fact that plenum-throttle models are present in experimental compressor rigs. The numerical modelling of a throttle-plenum configuration could be performed either explicitly, via the generation of a discretised domain, representative of the real experimental plenum-throttle configuration or implicitly, via a dynamic change of the boundary conditions at the physical outlet of the compressor in accordance to the output of a numerical 1D model.

The first, provides a more realistic modelling method since a more accurate representation of the shape of the plenum would enable the more precise capturing of both the direction and reflection of pressure waves inside the plenum. These are known to affect stall inception and the dynamics of the surge event [93]. In practice, precise modelling of the experimental rig is not feasible due to the complex ducting and the presence of other components that cannot be modelled. In several occasions, details regarding the geometry and the size of the plenum are not provided, thus accurate modelling of the plenum might not be possible. This modelling approach substantially increases computational requirements while introducing a considerable degree of complexity, both in terms of meshing as well as in terms of boundary conditions, especially if the throttle area varies during the simulation.

One-dimensional, throttle-plenum models were initially introduced by Greitzer [72] and provide a cost-efficient modelling alternative, whereby the entire plenum-throttle system is numerically represented via an 1D model, governed by a set of equations. While such models are incapable of accurately modelling phenomena related to the geometry of the plenum-throttle configuration, they provide extended flexibility since both the plenum volume and the throttle can be modified instantly and in a transient manner (throttle changes). Due to the flexibility and numerical efficiency it provides, the utilization of 1D model was deemed adequate for the context of the present work.

The plenum-throttle model used within the present work is the one proposed by Righi [30], whereby the flow-field solution of the plenum-throttle model is performed transiently by integrating in space and time the continuity and energy equation over a single-notional cell, attached to the physical outlet of the domain, the solution of which provides the boundary conditions. The plenum volume is considered large enough so that the flow inside is stagnant with the variations in the plenum's pressure and temperature being a function of the mass-flow which in turn is dictated by the throttle. The flow-field properties inside the plenum are obtained transiently by solving the following system of equations, where the subscripts Pl and T refer to plenum and throttle quantities respectively:

Continuity

$$\frac{\partial \rho_{Pl}}{\partial t} = \frac{1}{V_{Pl}} \left(\sum (\rho u_n A)|_{outlet} + \dot{m}_T \right) \quad (3.246)$$

Energy

$$\frac{\partial \rho_{Pl} E_{Pl}}{\partial t} = \frac{1}{V_{Pl}} \left(\sum (\rho u_n A H_0)|_{outlet} + \dot{m}_T c_p T_{0Pl,abs} \right) \quad (3.247)$$

Equations (3.246) and (3.247) essentially state, that the mass and energy flux into the notional plenum-throttle model, which numerically is treated as a single finite volume cell attached to the compressor duct outlet boundary, is equal to the summation of mass and energy fluxes through all finite volume cell faces adjacent to the outlet boundary. The throttle is treated as a simple convergent nozzle, the throat area of which, A_T , regulates the mass flow capacity of the system. To assess whether the nozzle is choked, isentropic relations are used:

$$Ma_T = \begin{cases} \sqrt{\frac{2}{\gamma-1} \left(\left(\frac{P_{Pl}}{P_{atm}} \right)^{\frac{\gamma-1}{\gamma}} - 1 \right)} & , \text{if } \frac{P_{Pl}}{P_{atm}} \leq \left(\frac{\gamma+1}{2} \right)^{\frac{\gamma}{\gamma-1}} \\ 1 & , \text{if } \frac{P_{Pl}}{P_{atm}} > \left(\frac{\gamma+1}{2} \right)^{\frac{\gamma}{\gamma-1}} \end{cases} \quad (3.248)$$

The mass flow at the throttle is subsequently determined using isentropic relations (eqn.(3.249)). The throat area, A_T is prescribed by the user and can be changed transiently. The equations are integrated over the notional finite volume cell (the plenum volume), while the time derivative terms are solved using a 4th order Runge-Kutta method.

$$\dot{m}_T = \sqrt{\gamma \rho_{Pl} P_{Pl}} A_T M_T \left(1 + \frac{\gamma-1}{2} M_T^2 \right)^{-\frac{\gamma+1}{2\gamma-2}} \quad (3.249)$$

3.9 Chapter summary

In this chapter, the theory and the modelling approaches underpinning the developed through-flow method are discussed. The generation of the grid is performed in a modular way, by connecting subdomains with user-specified resolutions. Regions with variable node distributions are optionally added to gradually match the cell size of adjacent domains. Bladed domains require a large number of streamwise elements ($\sim 80+$) to enable gradual flow turning and prevent numerical instabilities. Distinct pressure, suction and camber surface grids are generated for each blade using piecewise interpolation. The curvature of the blade, defined by the blade, flow path and lean angles is determined using a developed, vector-based method on the nodes of the camber surface grid. The blockage distribution is computed using the pressure and suction side grids. The extracted curvature and blockage distributions are subsequently interpolated onto the main computational grid using bilinear interpolation. The flow-field solution is obtained by solving the axisymmetric, inviscid Euler equations. Conceptual modifications in the governing equations are introduced to account for blockage effects, body forces and mixing effects. Depending on the component (duct, rotor or stator) the equations are modified to ensure consistency with the modelling assumptions. The solution is obtained in an empty duct without discretised blade geometries and losses and flow turning are introduced via body force-fields. The governing equations with blockage are derived in the relative FoR. A new, implicit approach for the evaluation of the additional blockage gradient terms is proposed which is not subject to the limitations of other methods reported in the literature. Shear stresses in the spanwise and streamwise direction are added in the governing equations to introduce a mixing mechanism. A density-based, finite volume solver is developed, incorporating the first-order accurate Godunov scheme, coupled with 3 Riemann solvers (HLL, HLLC, Roe-Pike) to compute the fluxes. The Riemann solvers are reformulated to ensure applicability with the relative velocity formulation of the governing equations. A new, explicit turning force model, applicable to all types of blade geometries is derived, following the principal ideas of [85]. A new correction force model is developed, capable of preserving the imposed flow direction at all flow conditions while being substantially more effective in impeller and vaned diffuser blades than Gong's [86]. A new, analytical viscous force model, applicable to all types of blades is developed, expressing the viscous force as a function of total pressure loss, flow path angle and local flow-field variables. The concept of CFD-derived distribution coefficients is subsequently introduced. Different methods

for the extraction and synthesis of those coefficients are proposed and introduced in the turning and viscous force models to distribute deviation and losses in a more realistic manner. A new, simplified, analytical blockage-mixing model is developed for the prediction of aerodynamic blockage, mixing losses and deviation due to mixing-out of the separated and jet flow at reverse flow conditions. The model is dissected into a constant area-mixing variation and a variable area-mixing one to ensure applicability to axial blades, impellers and vaned diffusers. The dedicated, component-specific loss and deviation models are subsequently presented. A shape function is introduced which essentially introduces a modified curvature region to gradually turn the flow. Depending on the flow regime inside a blade passage, different modelling strategies are adopted. Losses, deviation and aerodynamic blockage are predicted by the empirical models strictly at forward flow conditions. At reverse flow conditions, additional mixing losses, aerodynamic blockage and deviation are predicted by the analytical model while losses in the blade passage are introduced via the turning force. Except for passage losses, no blockage, mixing and deviation are introduced during mixed flow regime. For transient, post-stall aerodynamic simulations, an 1D throttle plenum model, proposed by Righi [30] is used.

Chapter 4

Methods - Validity and limitations

In this chapter, the validity and limitations of the developed modelling approaches, discussed in chapter(3), are investigated extensively, by carrying-out numerical-response test cases and utilizing CFD and experimental data. The solver is validated in a three-fold manner; a comparison and validation of the implemented Riemann solvers (HLL, HLLC and Roe) is performed against the analytical solution of Sod-shock type problems, the 2D axisymmetric solver is compared in an empty duct configuration against CFD results and finally test-cases are performed to evaluate the response of the solver with blockage effects and rotating domains. Potential numerical errors, induced by blockage modelling are discussed and methods to address them are proposed. The turning force model is validated in 5 compressor blade geometries including axial rotors, stators, impellers and vaned diffuser blades by comparing the discrepancies between imposed and actual flow angles. The viscous force is subsequently validated by comparing the imposed and actual, averaged pressure losses in an axial rotor and an impeller blade passage. The distribution coefficient concept is assessed by comparing the averaged, measured flow angle and pressure losses downstream the LE against the prescribed ones when different distribution coefficients are applied. Pressure and efficiency characteristics using different distribution coefficients are also obtained. The blade curvature evaluation approach is compared against the blade, lean and flow path angle distributions of rotor 37 blade, reported in the literature, while the blade angle distribution of impeller SRV2 is validated against experimental data. The validity of the new blockage-mixing model is subsequently assessed. The aerodynamic blockage predictions obtained from the new model at reverse flow conditions, are compared against the predictions of other models in the literature. Mixing losses and changes in flow angle due to mixing of the flow downstream blade passages are compared against CFD data. The losses inside the blade passage are extracted from CFD simulations

and compared against the losses imposed by the through-flow code via the turning force. The presence of large recirculation regions near the LE compromises the extraction of reliable data, thus the verification process is deemed preliminary. The chapter is concluded by investigating the effect of losses, deviation and aerodynamic blockage on the shape and position of the pressure and torque reverse flow characteristics.

Contents

4.1 Solver verification	170
4.2 Turning force model verification	186
4.3 Viscous force model verification	196
4.4 Distribution coefficient verification	201
4.5 Validation of blade curvature definition method	213
4.6 Reverse flow modelling - Validity and limitations	218
4.7 Blockage-mixing modelling implications	235
4.8 Chapter summary	243

4.1 Solver verification

The solution of the flow-field is performed internally, via a custom-developed solver embedded in the code, solving the modified 2D, compressible, axisymmetric Euler equations with blockage, as discussed in the methodology section (chapter(3)), in both the relative and the absolute FOR. Due to the inherently compressible, transonic, swirling flows encountered in axial and centrifugal compressors, the solver incorporates numerical schemes capable of robustly resolving transonic and supersonic flows. The governing equations are solved using the finite-volume, numerical scheme of Godunov, coupled with three approximate Riemann solvers due to its competent, shock-capturing capabilities [108]. The scheme requires the solution of the local Riemann problem to obtain the fluxes at each finite volume cell face and is first-order accurate in space and time. The accuracy of the scheme is also dependent upon the precision of the Riemann problem solution itself.

4.1.1 Verification and comparison of Riemann solvers

Three of the most widely used approximate Riemann solvers (HLL, HLLC and Roe-Pike) along with an exact one, are compared and their implementation is subsequently validated against the analytical solution of different Sod shock tube problems proposed by Toro [108]. Four test cases are simulated which essentially are variations of the classic Sod shock tube problem. Air is contained in 2 different states in a tube, separated by a diaphragm. Once the diaphragm is removed and depending on the conditions of the initial left and right state, different wave structures form. The table below summarises the initial conditions of the left and right states corresponding to the four different tests. The subscripts L and R denote left and right state properties respectively.

Test	ρ_L	u_L	P_L	ρ_R	u_R	P_R
1	1.0	0.0	1.0	0.125	0.0	0.1
2	1.0	-2.0	0.4	1.0	2.0	0.4
3	1.0	0.0	1000.0	1.0	0.0	0.01
4	1.0	0.0	0.01	1.0	0.0	100.0

Table 4.1: Initial conditions at the left and right states of the 4 Riemann problem tests.

Figure(4.1) shows the distribution of density, velocity, pressure and internal energy along the axial coordinate of the first test case at time $t = 0.25$ units. This is the classic Sod test problem and the wave pattern comprises a left rarefaction, a contact discontinuity and a right shock. The different types of waves can be identified by examining the velocity contours; in expansion fan regions velocity increases, in contract discontinuity regions velocity remains constant, while in shock wave regions decreases. The diffusivity of the scheme is evident by the smearing of the solution in the vicinity of regions where abrupt changes in the gradients of the displayed properties are present. The predictions of the HLLC and Roe's Riemann solvers are similar. Roe's solver is slightly more accurate in the region in-between the left rarefaction and the contact discontinuity ($x = [0 - 0.5]$), while the HLLC provides slightly better estimations in the shock wave region ($x = [0.9 - 1]$). The exact Riemann solver is the most precise one, while the HLL exhibits the most diffusive behaviour, resulting in more erroneous predictions compared to the rest of the Riemann solvers.

The second test consists of a two-rarefaction wave pattern with a contact discontinuity in the middle. The pressure in the contact discontinuity is nearly zero and can potentially result in divergence of the iterative scheme which determines the star properties (p^*, ρ^*) . This limitation was encountered in the Roe-Pike Riemann

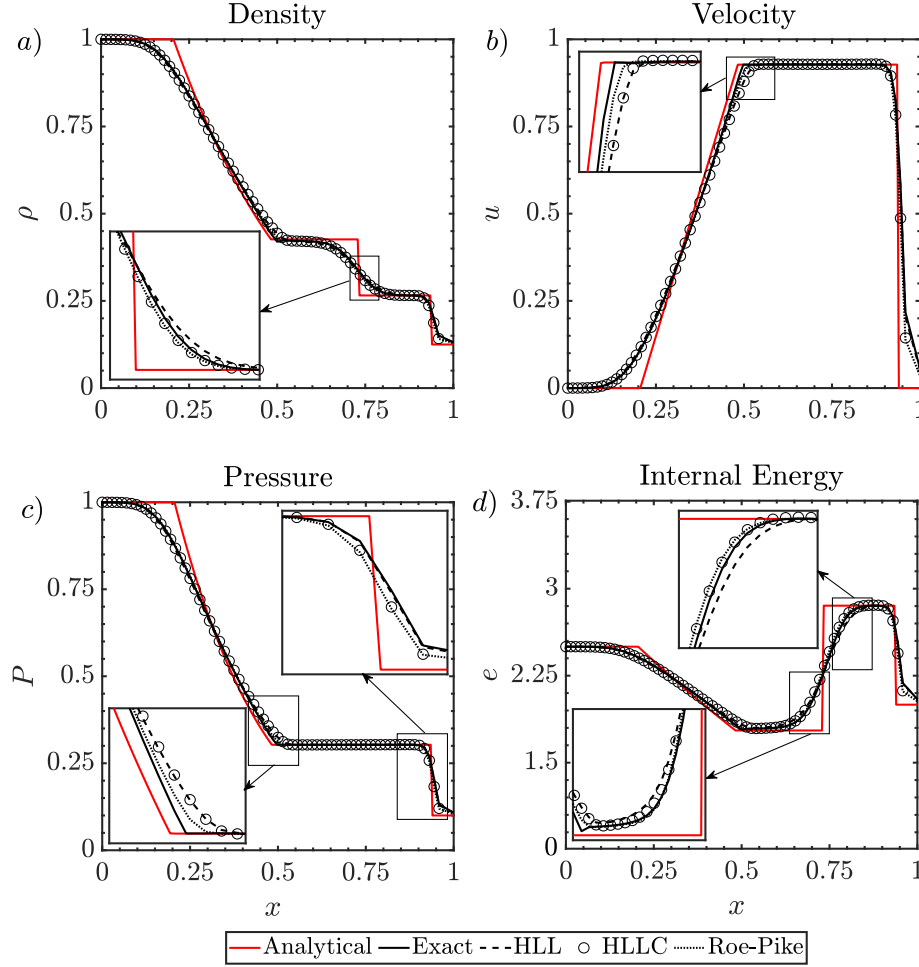


Figure 4.1: Comparison between analytical solution and different Riemann solvers of the first Sod shock tube problem at time $t = 0.25$ units.

solver, which was unable to resolve the wave structure due to misestimation of the star quantities (p^* , ρ^*). The same scheme is used for the determination of the star properties in all Riemann solvers. The excessively low pressure values in the contact discontinuity region magnify local misestimations of the primitive variables which are evident predominantly in the internal energy distribution graph (figure(4.2)) which in turn, is a function of the pressure and the density. The presence of the trivial contact discontinuity exacerbates the errors of approximate Riemann solvers with respect to the exact one, where the exact Riemann solver achieves substantial accuracy benefits. Density predictions favour HLLC over HLL solver, but the HLL provides more precise internal energy predictions in the contact discontinuity region ($x = 0.5$). It should be noted that this constitutes a rather distinctive, extreme test case and the errors and instabilities are not representative of most, real-life applications. This was demonstrated in the first test case (figure(4.1)) and will be further demonstrated in the next ones, presented hereafter. Numerical instabilities in numerous, complex

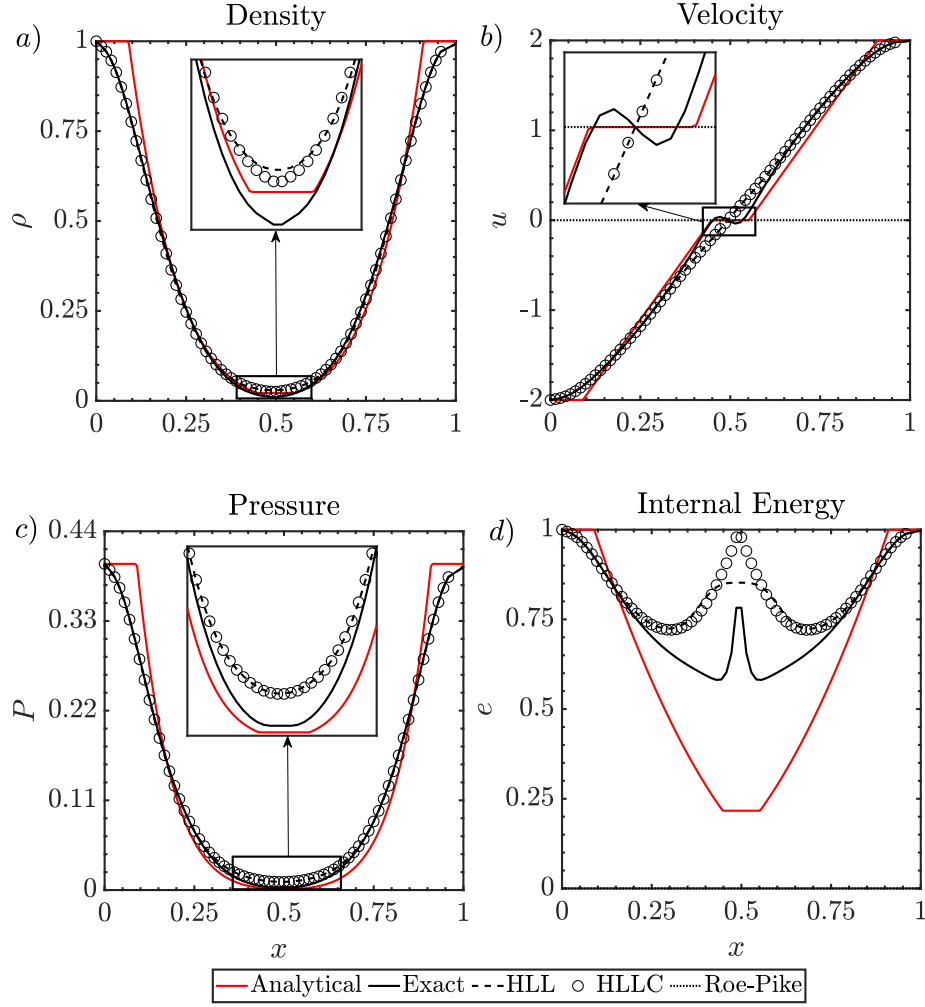


Figure 4.2: Comparison between analytical solution and different Riemann solvers of the second Sod shock tube problem at time $t = 0.15$ units.

body-force simulations have never been encountered with the current implementation of the Roe-Pike Riemann solver, unlike this particular test case might suggest.

Figures(4.3) and (4.4) depict the results corresponding to test 3 and 4, which consist of a similar but inverted wave structure; the first comprises a left rarefaction, a contact discontinuity and a right shock, while the latter consists of a wave structure, where the position of the rarefaction and the shock wave is inverted. The numerical diffusivity of the Godunov scheme is particularly noticeable in the density distributions graphs, in the vicinity of the contact discontinuity region where the smearing of the solution is excessively pronounced. The predictions of the other primitive variables obtained by all Riemann solvers correlate well with each other and are in very good agreement with the analytical solution. Compared to all approximate Riemann solvers, the exact Riemann solver correlates better with the exact solution but the accuracy benefits are rather minuscule. The limitations of the less sophisticated

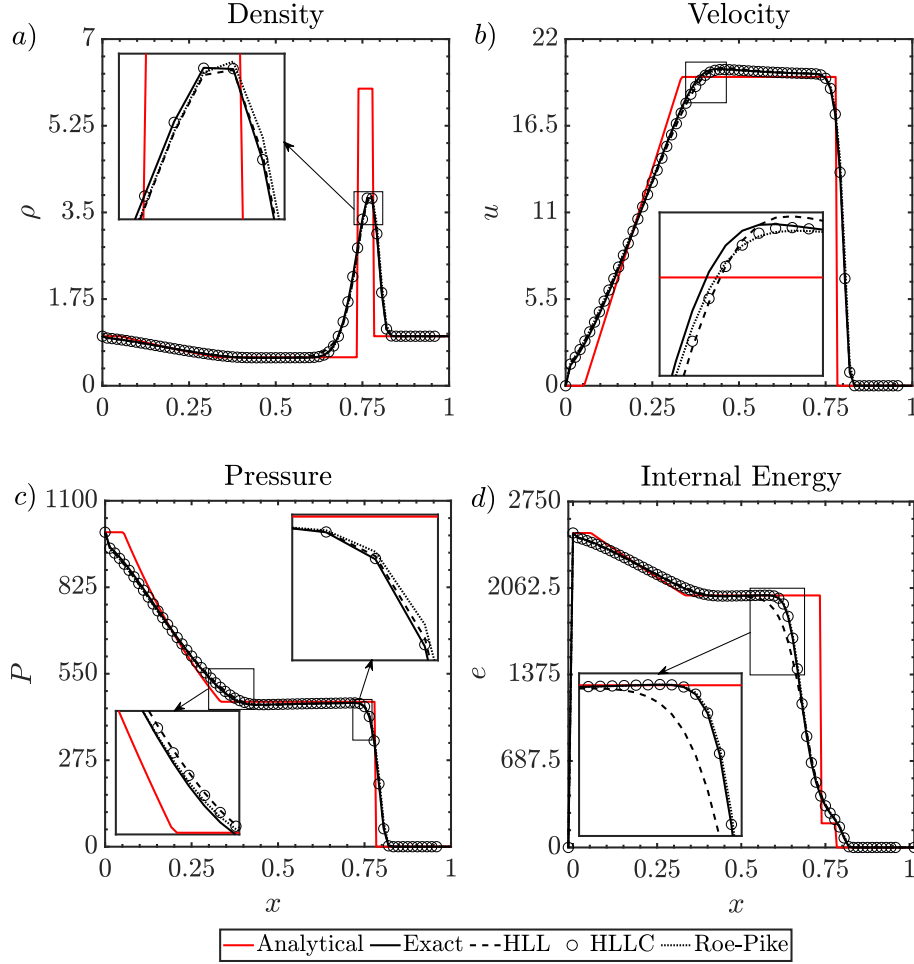


Figure 4.3: Comparison between analytical solution and different Riemann solvers of the third Sod shock tube problem at time $t = 0.012$ units.

HLL solver are more prominent in both tests, especially in the internal energy plots (figures(4.3,d) and (4.4,d)).

The comparison between approximate and exact Riemann solvers against the analytical solution of different Sod shock tube problems validates the correct implementation of the Riemann solvers and facilitates the deduction of certain aspects underpinning Godunov's scheme, regarding their capabilities and limitations. The diffusivity of the scheme is evident whenever abrupt changes in the gradients of the flow-field manifest as a consequence of different pressure wave structures. Despite the high numerical diffusivity of the scheme, its monotonicity-preserving characteristics, make it very robust without requiring flux limiters to preserve numerical stability. The exact Riemann solver constitutes the most accurate solver, but the accuracy benefits are rather insignificant compared to more sophisticated approximate Riemann solvers, such as the HLLC and Roe's and do not outweigh the associated numerical cost and stability downsides it implies. Exact Riemann solvers require the solution of

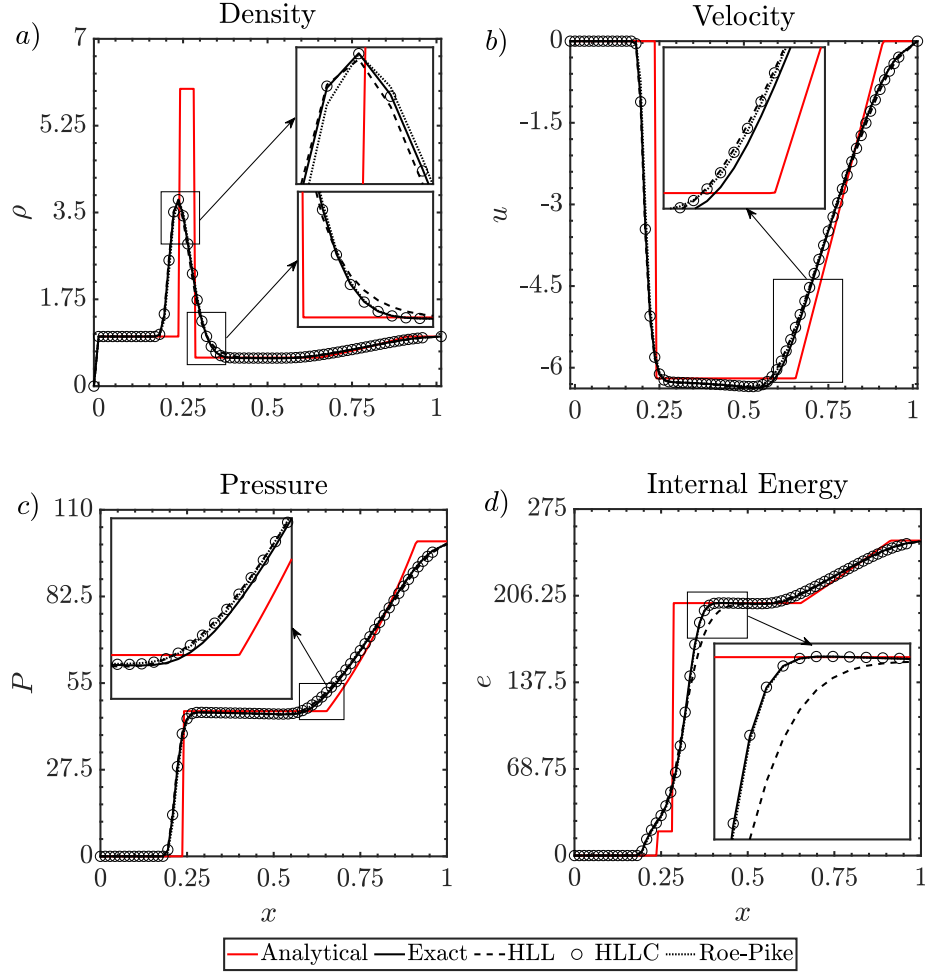


Figure 4.4: Comparison between analytical solution and different Riemann solvers of the forth Sod shock tube problem at time $t = 0.035$ units.

the pressure equation via an iterative scheme which not only introduces additional computational cost but also might compromise stability, since the solution of the scheme might diverge if the initial guess is highly inaccurate. The HLLC and Roe's Riemann solvers were deemed the most balanced ones, in terms of both numerical efficiency and accuracy and therefore are the principal Riemann solvers used in the code. The HLL Riemann solver is incorporated in the code due to its simplicity and numerical efficiency, but its usage is avoided due to inferior accuracy.

4.1.2 2D axisymmetric solver verification

The verification process of the solver is performed via a combination of test cases aiming to ensure the correct implementation of key-aspects of the solver; the axisymmetric implementation, the flux correction at interfaces adjoining domains with different rotational velocities and the precise evaluation of the blockage gradient

terms. The first, is performed in an axisymmetric centrifugal compressor duct, corresponding to the flow-path geometry of impeller SRV2 [107] without blades, blockage or body forces. Its objective is the verification the correct implementation of the 2D, axisymmetric, Euler equations. The aim of the second test is the assessment of the blockage gradient evaluation method using an adaptive mesh, representative of the impeller SRV2 duct with metal blockage, a rotating impeller domain but no body forces. The total inlet and static outlet pressure and the initial pressure-field along the duct is the same; this should result in a totally stagnant, invariant, uniform flow-field and any changes would indicate misestimations of the blockage gradient terms. The last test is performed using the same computational domain and set-up, but a pressure gradient is applied along the duct; using the solver in the absolute and the relative FoR, a qualitative flow-field comparison along with mass flow and mass-flow-averaged total pressure and temperature ratios is performed to ensure mass flow and energy conservation and verify the validity of the flux-correction method used in the relative FoR solver.

Solver verification - 2D Axisymetric without blockage

The discretisation of the blade geometry and the solution of the flow-field is initially performed using the in-built mesh generation algorithm and the developed solver; a single-zone grid, comprising 200×50 streamwise and spanwise elements respectively constitutes the computational domain, while the steady-state flow-field solution is obtained in a transient manner using the HLLC approximate Riemann solver.

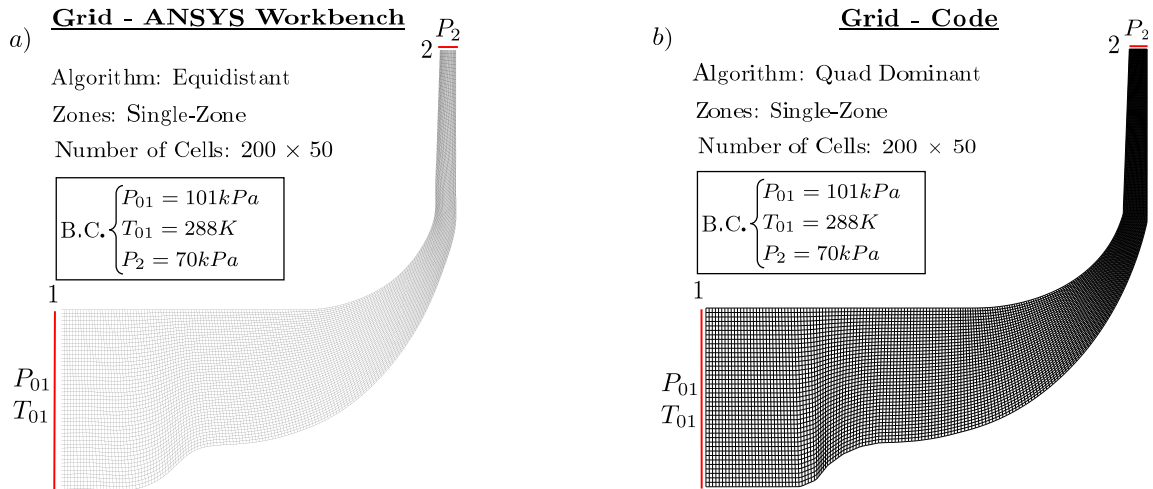


Figure 4.5: Computational domain of centrifugal compressor SRV2-0 duct generated by a) the code and b) ANSYS FLUENT mesher.

Blockage is not modelled and all domains are stationary. A similar single-zone grid, with the same element count is generated in ANSYS mesher. The flow-field is obtained using a first-order accurate, density-based solver using the Roe-FDS scheme to obtain the fluxes and the Green-Gauss, cell-based method to compute the gradients. Stagnation temperature and pressure boundary conditions are specified at the inlet, while static pressure is imposed at the outlet. The computational grids and imposed boundary conditions of the through-flow and the ANSYS models are depicted in figure(4.5)

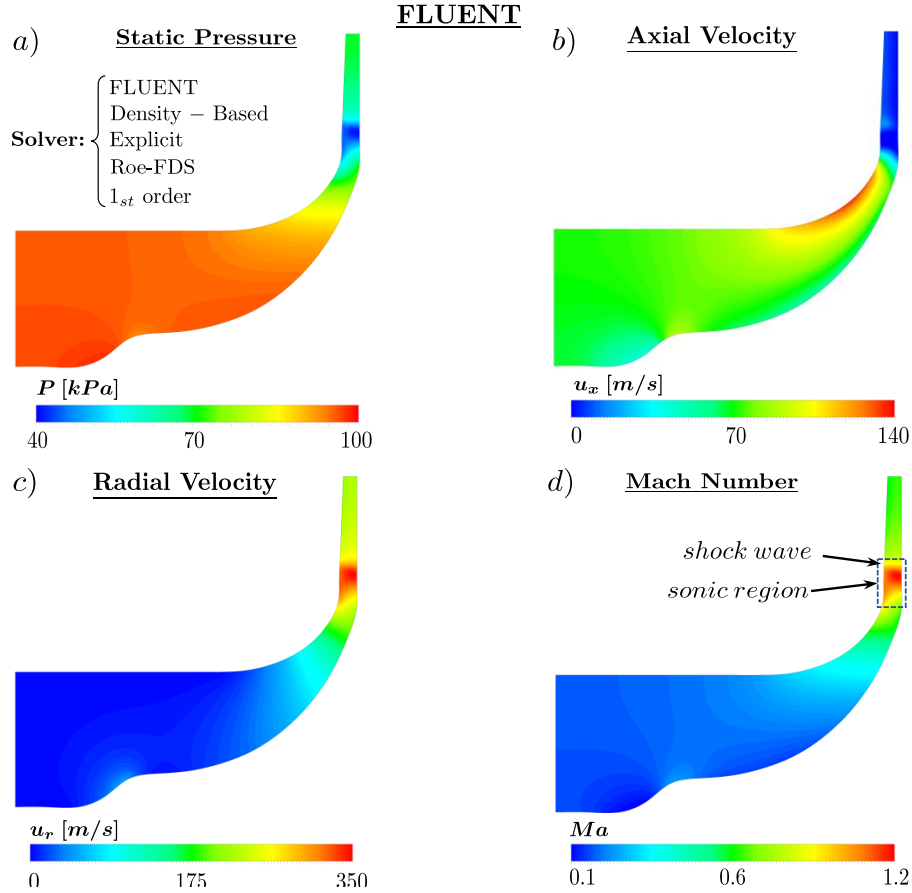


Figure 4.6: a) Pressure, b) axial velocity, c) radial velocity and d) Mach number contours along the compressor SRV2-0 duct, obtained from FLUENT.

Figures(4.6) and (4.7) show the flow-field solution obtained from FLUENT and the custom solver respectively. The contours show static pressure, axial velocity, radial velocity and Mach number distributions along the duct. The stagnation inlet properties are set to standard ISA ones, i.e. $P_{01} = 101kPa$, $T_{01} = 288K$, while the static pressure at the outlet is set to $P_2 = 70kPa$, resulting in a duct pressure ratio of $PR_{duct} \approx 0.69$. The pressure ratio is set high enough to ensure the formation of shock waves and allow a direct assessment of the shock-capturing capabilities of the

developed solver. The Mach number contours indicate the extent of the super-sonic region at the inlet of the vaneless diffuser, followed by a shock-wave which leads to an increase in the static pressure and decrease in the local velocity. This is successfully reproduced by the developed solver and the HLLC Riemann solver, with the extent of the supersonic region being slightly larger. Minor differences can be seen at the hub line, near the inlet bell-mouth curvature area and in the axial velocity contours at the inlet of the vaneless diffuser, but these are presumably related to differences in the local hub and shroud curvature, since these are generated using different interpolation schemes. The flow-field solution obtained with the custom solver is in very good agreement with the one of FLUENT, verifying the correct implementation of the numerical solver and confirming the accuracy and shock-capturing capabilities of the solver.

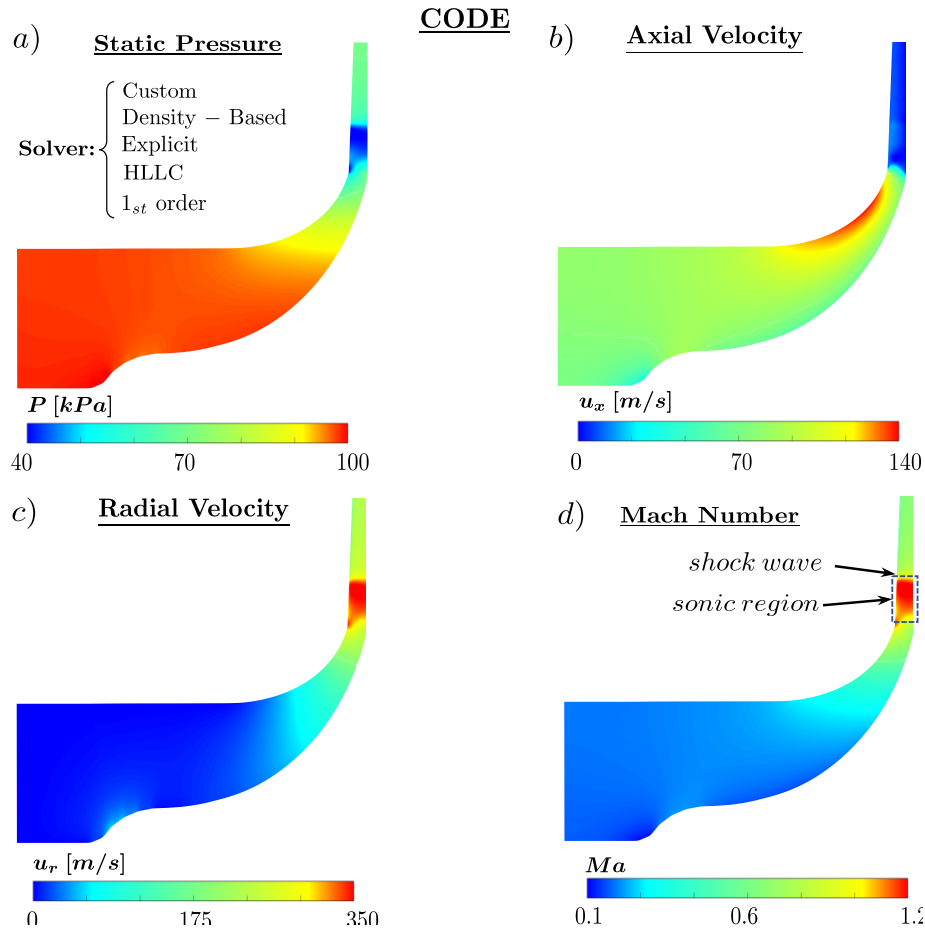


Figure 4.7: a) Pressure, b) axial velocity, c) radial velocity and d) Mach number contours along the compressor SRV2-0 duct, derived from the custom solver.

Solver verification - 2D Axisymmetric with blockage

The second test case aims to assess the precision of the developed blockage gradient evaluation approach and provide an initial verification step of the flux correction method at interfaces adjoining domains with different angular velocities. The precise evaluation of the blockage gradient terms constitutes a vital numerical aspect of the solver, since even ostensibly minor misestimations of those terms was found to result in significant momentum imbalance, acting as body force source terms. The precise approximation of those gradients, particularly in the more complex impeller blade geometries, was found to be exceptionally challenging using finite difference approximations, thereby deeming their usage within the context of the present work inadequate.

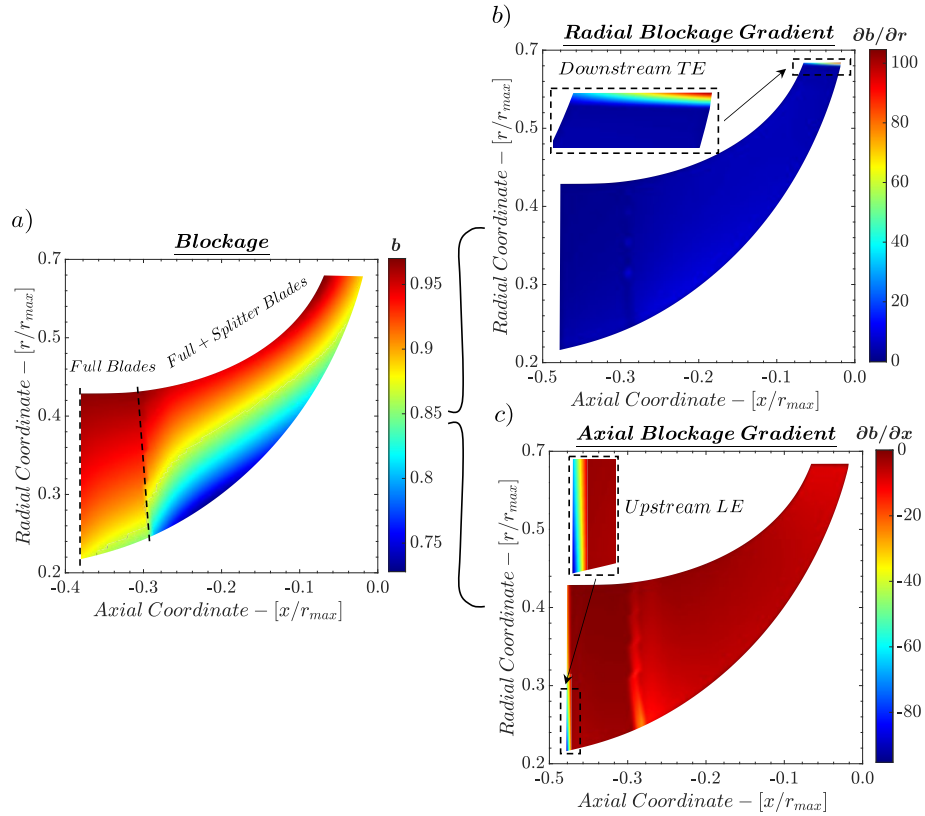


Figure 4.8: Distribution of Impeller SRV2 a) metal blockage, b) radial blockage gradient and c) axial blockage gradient. The presence of large blockage gradients at regions upstream and downstream the LE and TE respectively are highlighted in the zoomed regions.

Figure(4.8) shows the blockage distribution of impeller SRV2 along with the corresponding radial and axial metal blockage distributions. The presence of the splitter blade is reflected on the blockage distribution contours, which results in an increase in the local blockage, expressed via a decrement in the local blockage parameter values downstream the LE of the splitter blade. The blockage gradient

contours indicate the existence of large $\partial b/\partial x$ and $\partial b/\partial r$ values just upstream (axial gradient) and downstream (radial gradient) of the LE and TE respectively. These large values manifest as a result of the transition from a domain without metal blockage (duct) to a domain with metal blockage (blade passage). The abrupt reduction in the volume and areas of the cells at the LE and TE due to metal and aerodynamic blockage, can potentially induce numerical, mesh-related errors, thus explicit treatment is required (see section(4.1.3)).

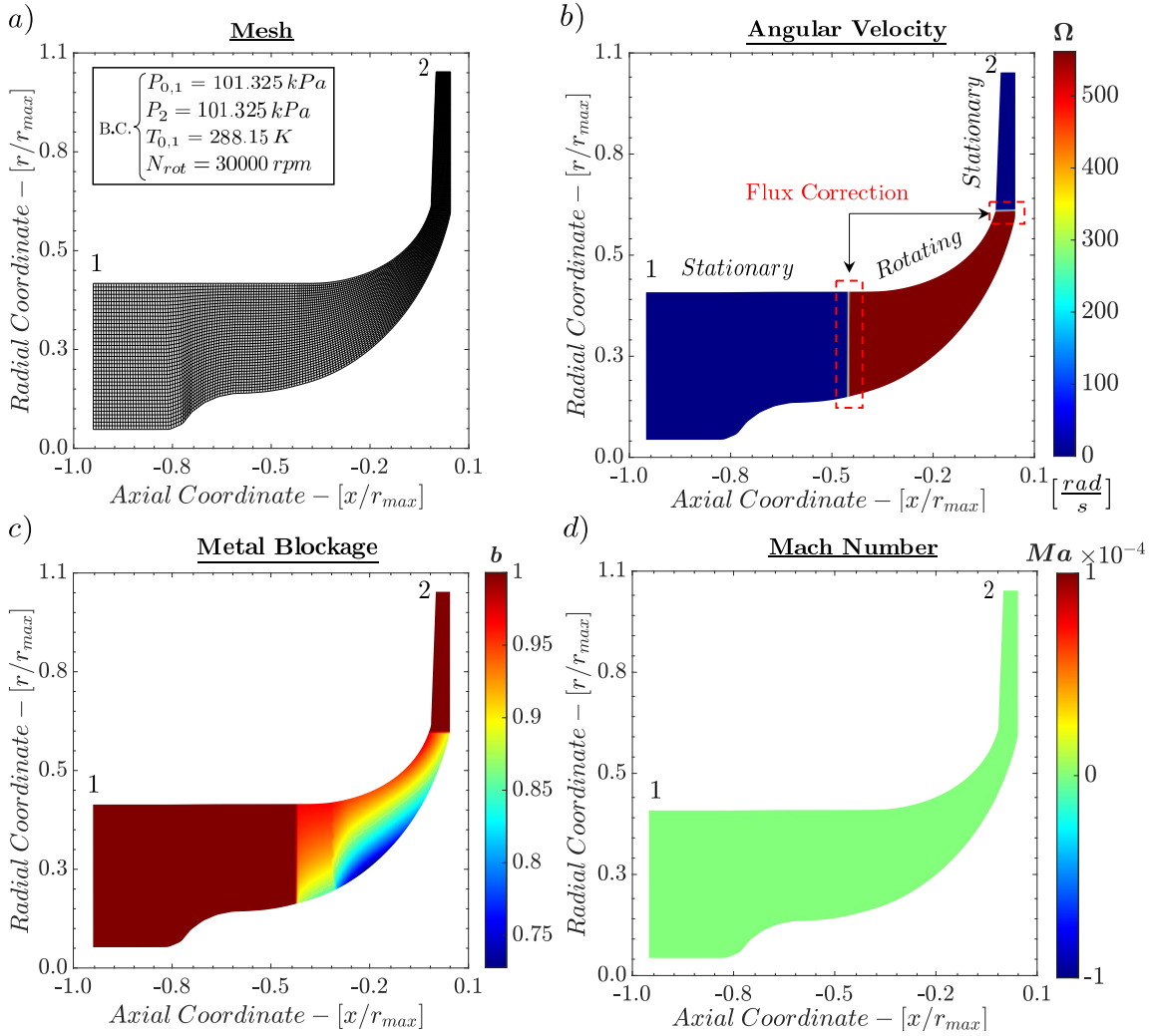


Figure 4.9: Impeller SRV2 duct with metal blockage, rotating and stationary domains and no body forces to the verify flux correction and blockage gradient evaluation methods.

To verify the validity of the blockage gradient evaluation method, the SRV2-0 duct is discretised following the adaptive mesh approach, thereby ensuring that the grid lines and the meridional projection of the impeller's LE and TE are precisely aligned (figure (4.9a)). Standard ISA ambient conditions are imposed at the inlet and the outlet to ensure a stagnant, invariable flow-field. Any variations in the flow-field

solution would then be strictly associated with errors in the blockage gradient terms, leading to axial and/or radial momentum imbalance. Metal blockage effects are considered in the analysis, but body forces are not exerted in the impeller domain (figure (4.9c)). The rotational speed is set to 30000rpm in the bladed domain and zero in the rest of the duct (figure (4.9b)). The Mach number contour shown in figure (4.9d) confirms that the flow-field remains stagnant and uniform, thus no momentum imbalance is introduced due to misestimation of the blockage gradient terms.

Solver verification - 2D Axisymmetric with blockage, relative FoR

The verification of the solver is concluded using the same computational domain and the blockage distribution of impeller SRV2 shown in figure(4.9). A pressure gradient is imposed along the duct by setting the static pressure boundary condition at the outlet to 95kPa. The bladed domain is rotating at 30000 rpm but no body forces are exerted on the cell centroids in the bladed domain. This aims to isolate the analysis from potential interference of the body-forces in the flow-field solution. The flow-field solution is obtained by the custom solver, using both the absolute and the relative velocity formulation of the governing equations with blockage. Since the solver in the absolute FoR does not require flux correction at rotating/stationary interfaces, its solution is used to assess the validity of the relative FoR solver's flux correction method. Mass momentum and energy conservation are subsequently evaluated by comparing mass flow, stagnation pressure and temperature ratios.

Figure(4.10) depicts the steady-state contours of the Mach number, the stagnation pressure and temperature using the developed solvers in the absolute and the relative FoR. The flow-field solutions of both solvers are identical, indicating the validity of the flux-correction method. The mass flow predicted by the relative FoR solver is overestimated by 0.04% while in both cases the mass flow-averaged total pressure ratio, a measure of losses, indicates a total pressure drop of 0.4% arguably related to the small separation region at the diffuser inlet, the numerical diffusivity of the scheme and potentially the lack of blockage smoothening at the LE and the TE (see section(4.1.3)). The total temperature ratio, a quantity representative of energy conservation along the duct, is in both cases close to unity, verifying energy conservation in both solvers when rotating domains and blockage are modelled.

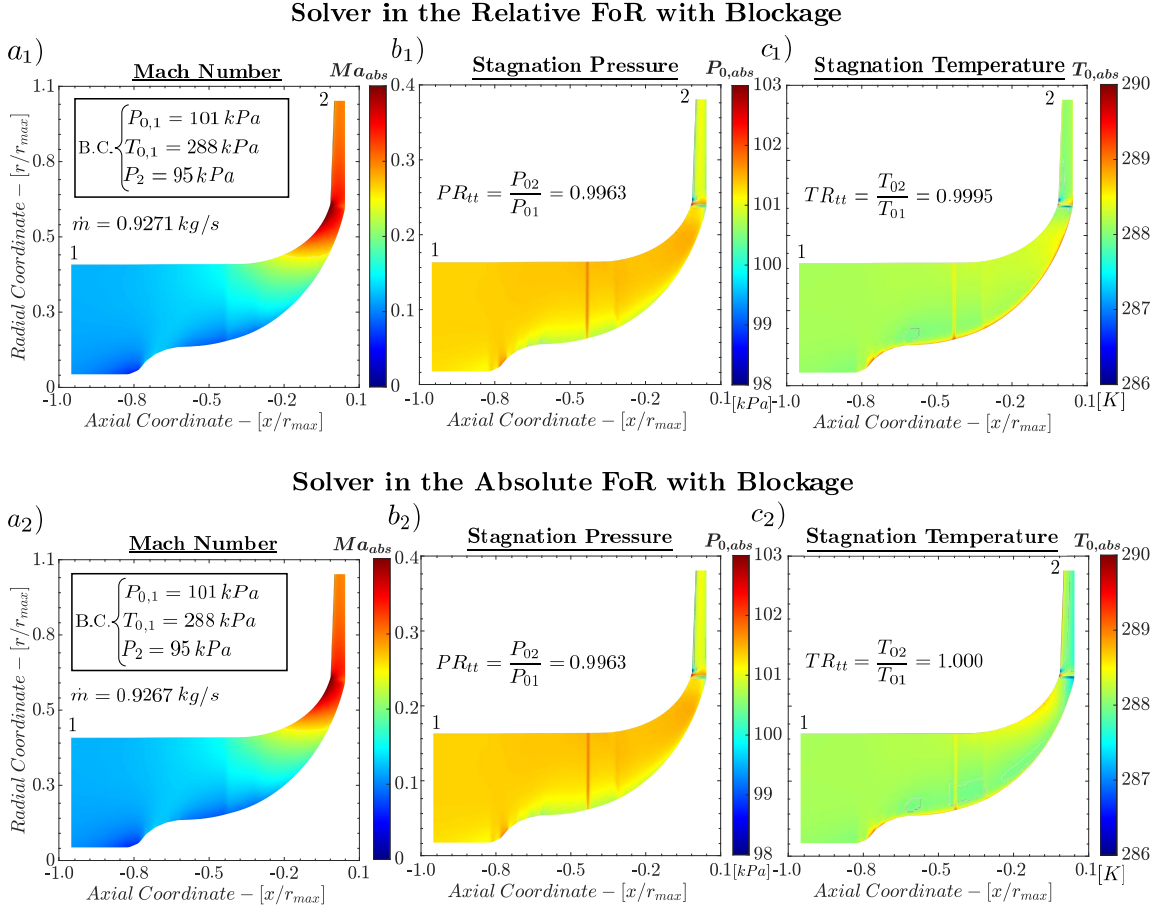


Figure 4.10: Mach number, stagnation pressure and temperature contours obtained from the custom solvers in the relative and absolute FoR.

4.1.3 Blockage modelling considerations

In the previous subsections, a series of test cases were performed, aiming to verify the correct implementation of the solver, emphasizing on certain aspects, in particular flux correction and the blockage gradient term evaluation approach. While the latter ensures the precise evaluation of the additional blockage-related terms, it does not constitute the sole source of numerical error when blockage is modelled. Abrupt, large changes in cell faces and volumes might lead to unrealistic flow-field features due to numerical errors. The source of these errors is identical to grid-related errors; the blockage parameter decreases the effective available area and volume, essentially modifying the effective computational domain. The aforementioned abrupt changes manifest at the LE and TE of bladed domains, where a sudden change in the cell face and volume occurs when the flow exits or enters bladed domains where blockage effects are modelled. This is not a major consideration when modelling axial compressor blades and impellers due to small metal blockage at the LE and

TE but it is crucial in vaned diffuser blades, especially wedge type ones, due to the excessively large thickness of the blades at the TE. This is further exacerbated due to the increase in the radius of the cells which results in large changes in cell areas and volumes even if blockage effects are entirely neglected. This modelling consideration is also consequential in reverse flow modelling due to large aerodynamic blockage.

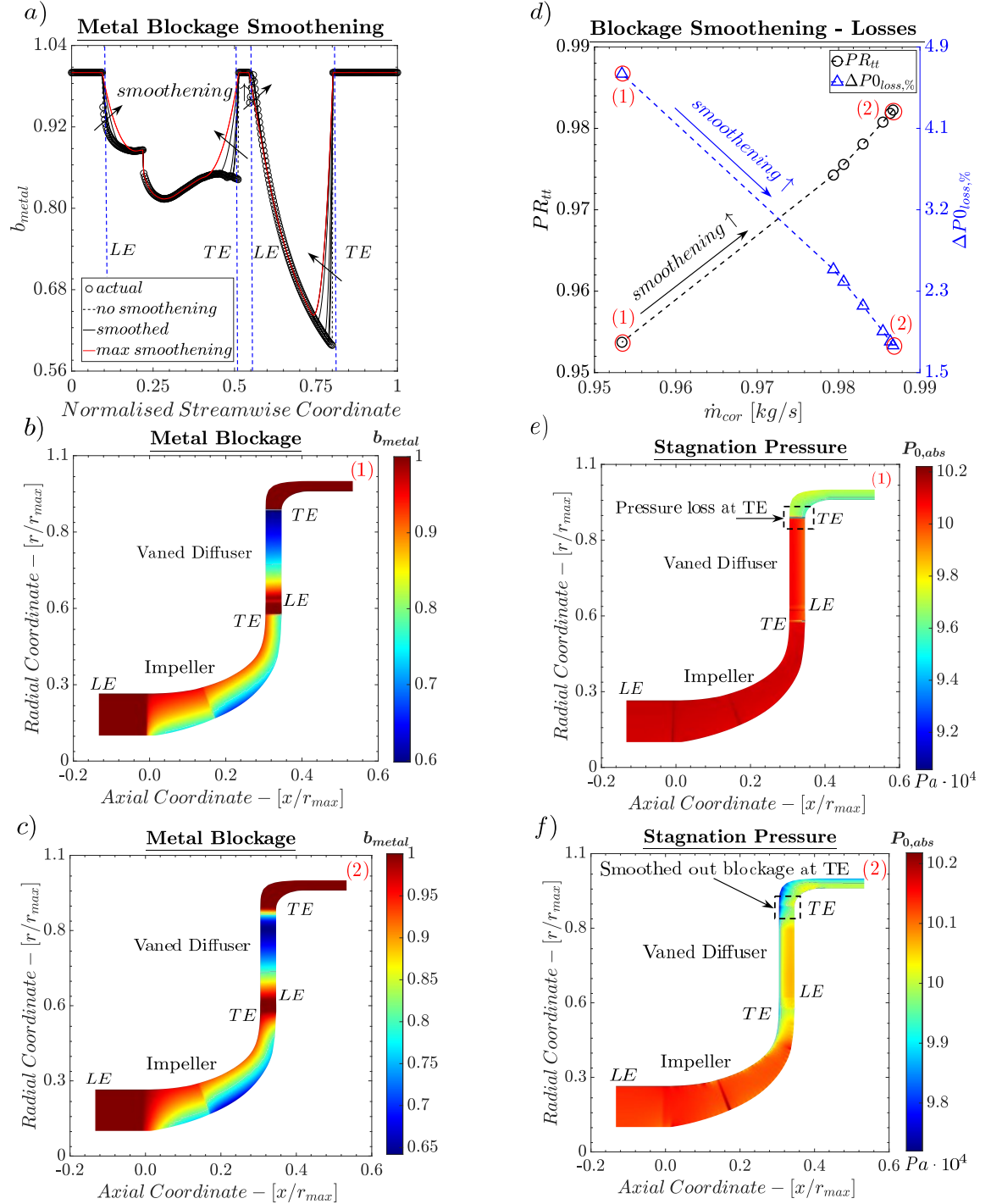


Figure 4.11: Blockage smoothing at the LE and TE of the NASA CC3 blades to reduce numerically induced pressure losses.

Figure(4.11a) shows the metal blockage distribution at midspan of NASA CC3 compressor [119], a centrifugal compressor comprising an impeller and a wedge-type, vaned diffuser. Using the custom cubic shape function presented in section(3.7.4), the blockage distribution is smoothened out, with the red line corresponding to the maximum smoothening case. In ducted regions, the blockage parameter is 1, while at the TE of the vaned diffuser ($m = 0.75$), the metal blockage reaches almost 40% ($b \approx 0.6$). The baseline, zero-smoothening blockage contours and the highest smoothened one, are shown in figures(4.11b,c). To investigate the implications of the baseline model and assess the effect of blockage smoothening, the NASA CC3 compressor is modelled as an empty duct with metal blockage, no body forces and a pressure gradient such to prevent the formation of shock waves. The mass-flow-averaged total pressure at the inlet and the outlet is compared to identify numerically-related losses due to blockage modelling. Figure(4.11d) shows the pressure ratio and the total pressure loss at the outlet for different degrees of smoothening. The baseline model results in a total pressure loss of around 4.8% (point 1), due to the sudden expansion of the flow downstream the TE of the vaned diffuser which in turn, precipitates a numerically-induced pressure loss and deceleration of the flow (figure(4.11e)). Introducing blockage smoothening at the LE and the TE of all blades mitigates this numerical error, preventing the sudden expansion of the flow and resulting in a much more realistic flow-field. The extent of the modified blockage region is adjusted in a way to prevent large transition regions which could potentially lead to erroneous blockage values at the throat and consequently misestimations of the choking mass flow. While total pressure losses are not entirely eliminated, they are reduced to acceptable levels which do not exceed 1.8%.

The aerodynamic blockage distribution along blade passages is smoothened out following a similar approach. To avoid adding excessive additional computational cost, a simple linear function is used to transiently modify the aerodynamic blockage distribution, instead of the custom cubic shape function used to modify the metal blockage one (figure(4.11a,c)). Test cases were carried out in a different centrifugal compressor duct (CSTAR). Following a similar approach, body forces and rotating domains are omitted, while aerodynamic blockage is imposed in impeller and vaned diffuser passages (no metal blockage). The extent of the linearly modified blockage region is automatically adjusted transiently via an algorithm ensuring that changes in blockage along the passage will not exceed a prescribed value. The ability of the aerodynamic blockage smoothening approach to prevent numerically-induced losses is evaluated at a range of different aerodynamic blockage levels along with the total

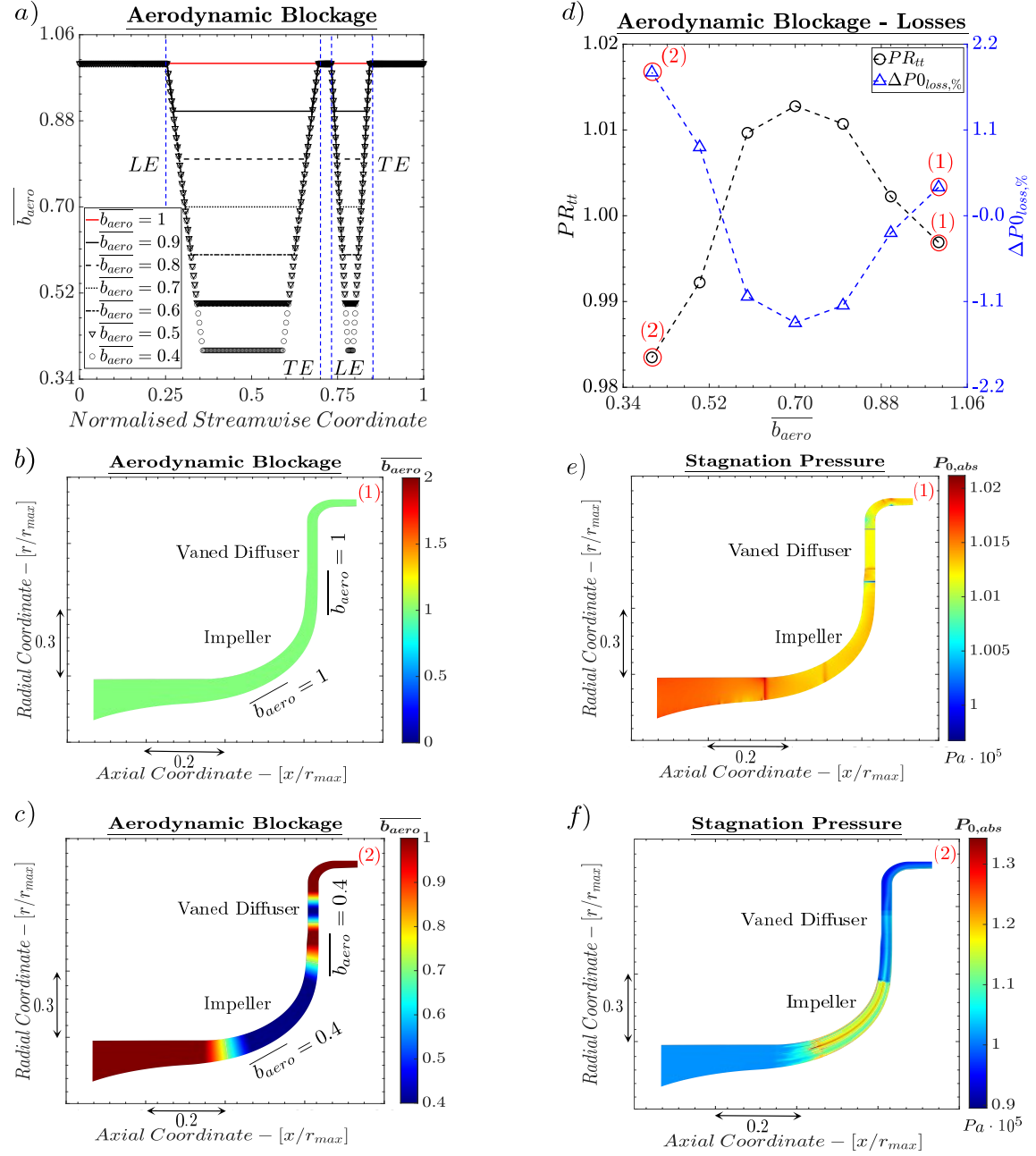


Figure 4.12: Reduction in mesh-related errors for different levels of blockage smoothing (aerodynamic) in CSTAR duct and overview of the blockage and total pressure contours with and without smoothing.

pressure ratio and the corresponding pressure drop (figure(4.12a,d)). Stagnation pressure variations in the higher blockage parameter range, $\overline{b}_{aero} = [0.7 - 1]$ are trivial, with the measured total pressure at the outlet being up to 1.4% higher than the inlet one ($\overline{b}_{aero} = 0.7$). The recorded pressure drop of up to 2% when the imposed aerodynamic blockage exceeds 30%, i.e. $\overline{b}_{aero} < 0.7$, occurs due to the formation of a shock wave upstream the impeller TE ($\overline{b}_{aero} = 0.4$). The location and the effect of

the shock wave is evident by examining the total pressure contour corresponding to the highest blockage setting (figure(4.12f)) manifesting as a discontinuous decrement in stagnation pressure. The outcomes of the test cases clearly indicate that at forward flow conditions, where aerodynamic blockage generally is within a 5%-10% range, as well as at reverse flow-conditions, where extensive separated regions and consequently substantial aerodynamic blockage is encountered, the associated amount of numerically-related errors is mitigated within reasonable limits.

4.2 Turning force model verification

The ability of the developed turning force model to precisely align the flow with the imposed relative flow-path is of paramount importance, both at transient and steady-state conditions, since it ultimately dictates the turning of the flow and consequently, the amount of energy input into the flow. While global compressor aerodynamic performance predictions are exclusively dependent upon the steady state of the flow-field (forward and reverse characteristics), transient compressor aerodynamic performance, e.g. rotating stall and surge are inherently transient phenomena. The ability of the turning force to transiently estimate the required force to achieve the imposed flow turning is therefore the most essential modelling aspect of body-force-based modelling approaches. Since the present turning force model is developed with the intention of universal applicability regardless of the type of blade under consideration, it was tested on five different compressor blade geometries: impeller blade (SRV2 [107]), vaned diffuser blade (NASA CC3 [119]), small axial fan rotor (rotor 67 [120]) and stator (stator 67 [120]) and an axial compressor rotor (rotor 37 [121]). The first 4 test cases are performed by imposing a prescribed amount of deviation but no losses, while no deviation was applied to the latter one, since it was used to demonstrate certain limitations of the correction force model.

Figure(4.13) shows the blade, imposed and actual flow angles along the hub, midspan and shroud of impeller SRV2. The modified curvature region in the vicinity of the LE is set to 5% of the blade's meridional length, and the flow misalignment is shown by the blue dashed line, measured as the percentage difference of the actual and the imposed relative flow angle. It is evident that the turning force model is capable of turning the flow with exceptional precision, with the maximum recorded misalignment being slightly above 0.5% at the shroud TE (figure(4.13d)). Precise flow guidance at the TE is essential, since it underpins the energy exchange between the fluid and the blades. Even though misalignments in the vicinity of the LE do

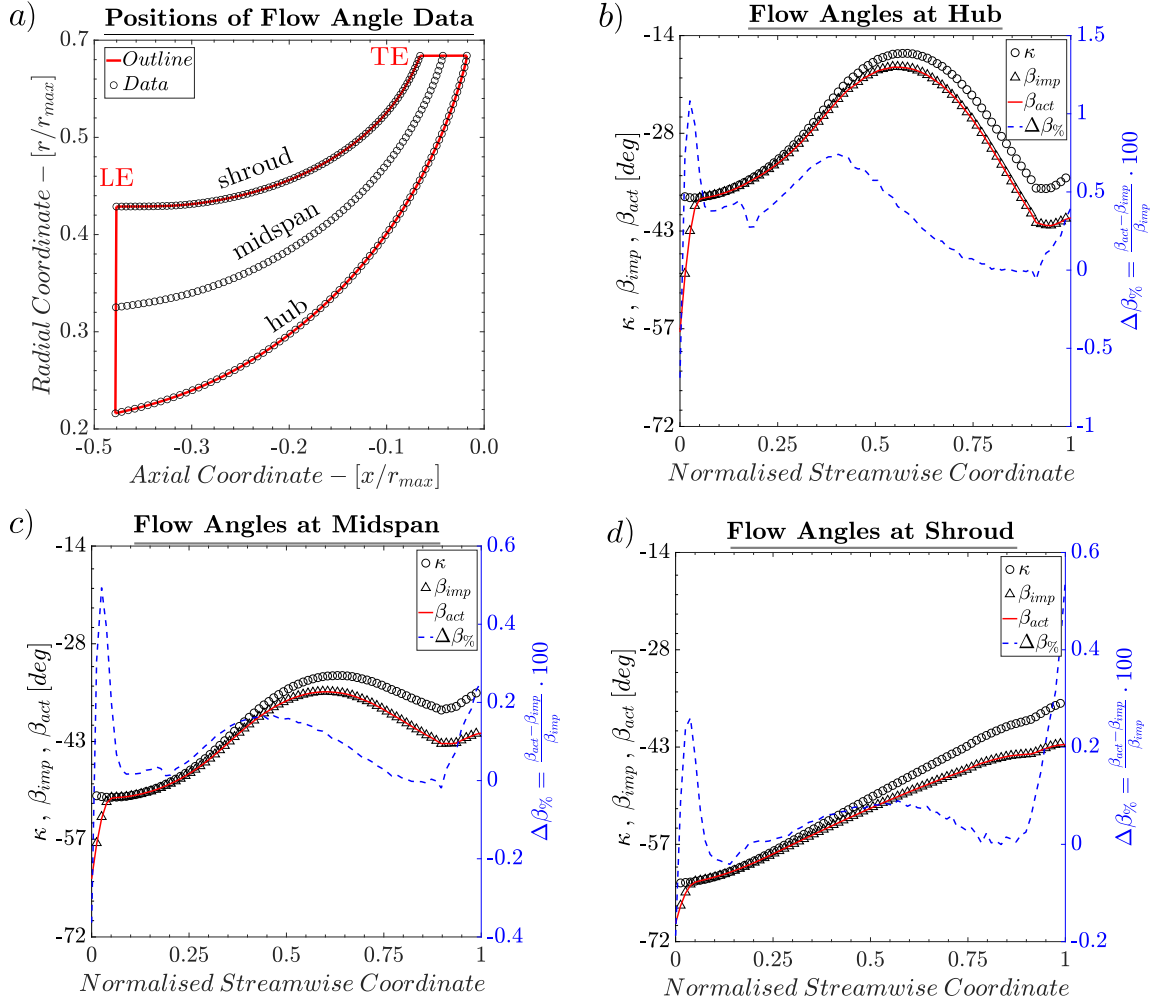


Figure 4.13: Blade, imposed and actual relative flow angle distributions at the hub, midspan and shroud in impeller SRV2 passage.

not affect the overall flow turning, flow misalignment near the throat might alter the choking capacity of the compressor. The errors in the actual flow path are of similar order to the ones near the TE, thus choking and energy exchange errors associated with the flow turning are expected to be trivial.

The decomposition of the turning force model into its constituent components, namely the base turning force and correction force is shown in figure(4.14). High turning forces are imposed by the base model at the LE, due to the abrupt flow turning of the incoming flow once it enters the bladed domain, as well as at the TE, due to changes in the blade curvature. These also are the regions where the interference of the correction force model is more pronounced, highlighting the significance of the correction force model. The high interference of the correction force near the TE justifies the usage of a different characteristic length in the correction force model, instead of the blade pitch used by Gong [86], since the increase in radius closer to

the TE as well as from hub to tip, would inevitably result in reduced flow correction and in turn, larger misalignments.

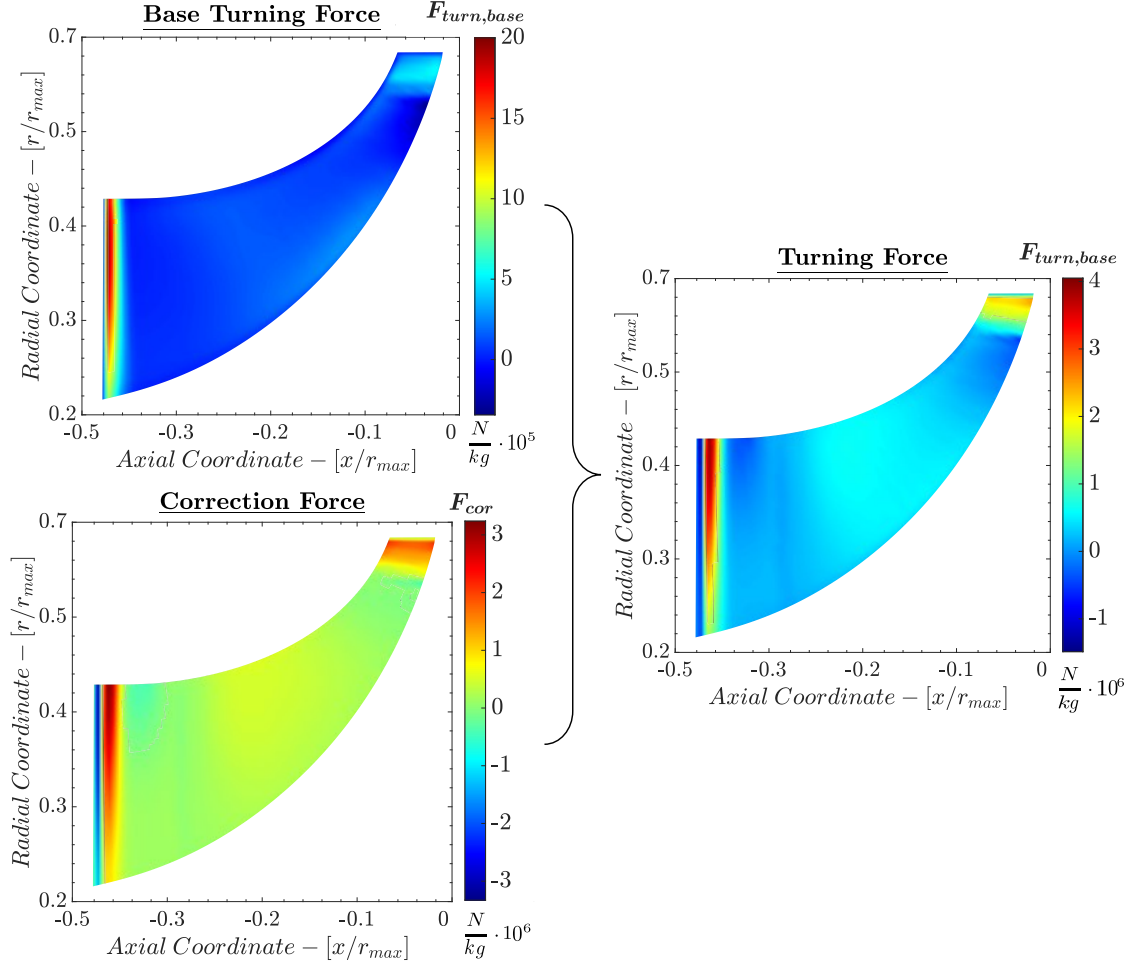


Figure 4.14: Base turning, correction and turning force contours in impeller SRV2 passage.

The turning force model was subsequently evaluated on the vaned diffuser blade of the NASA CC3 centrifugal compressor. The flow angles illustrated in figure (4.15) are essentially absolute flow angles since the blades are stationary. The imposed flow angle distribution is in excellent agreement with the imposed ones, with the largest misalignment located in the vicinity of the TE at the hub and midspan. In both cases the misalignment does not exceed 1.7%. Substantial flow misalignment near the throat may lead to over or underestimation of the choking mass flow in both rotors and stators. The errors near the LE, in the modified curvature region, are of the same order, thus they are not expected to introduce any errors in the choking capacity of the compressor.

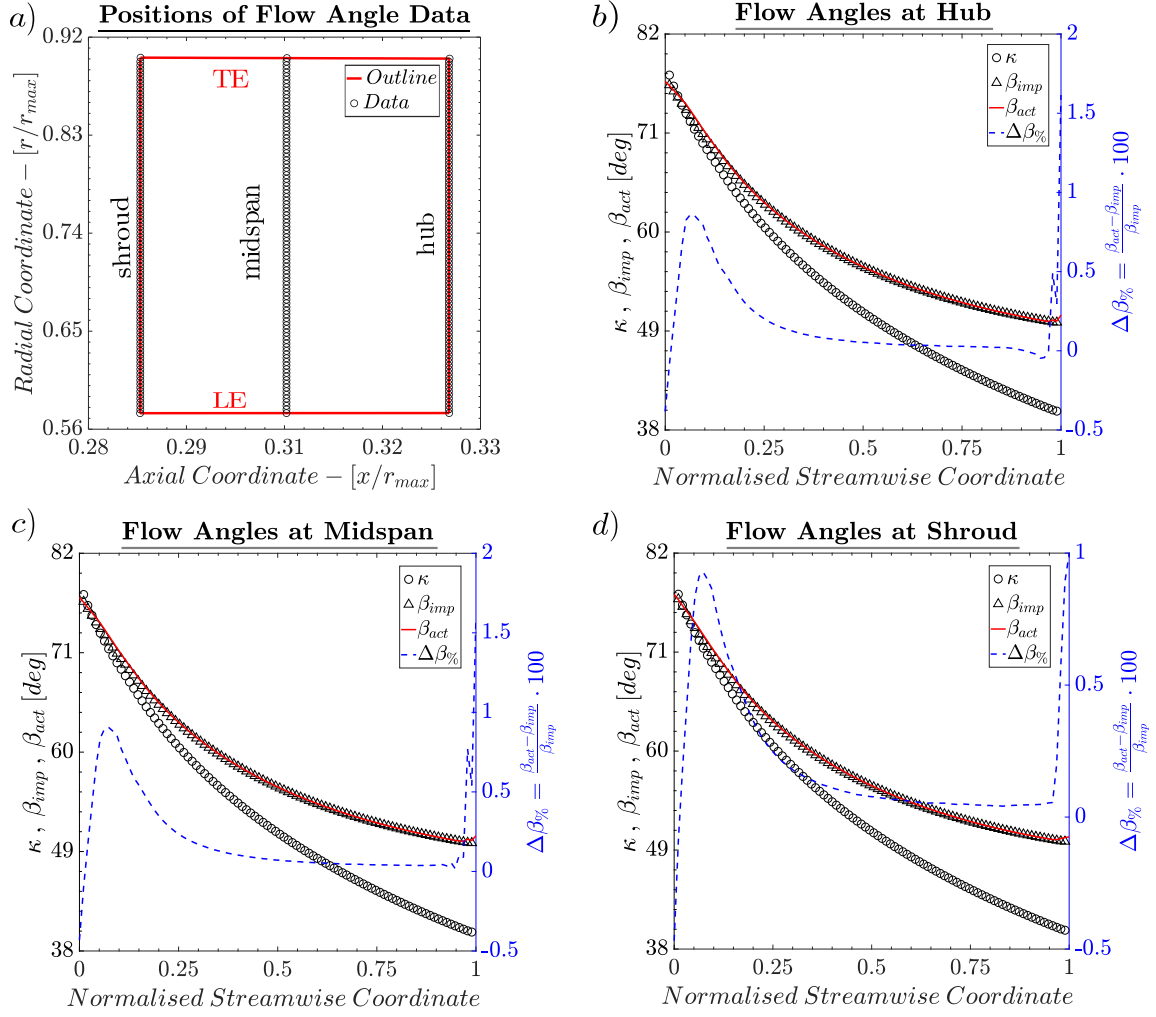


Figure 4.15: Blade, imposed and actual absolute flow angle distributions at the hub, midspan and shroud in NASA CC3 vaned diffuser passage.

Figure(4.16) shows the turning force and its constituent components. Large base and correction force values are recorded in the LE region in an attempt of the turning force model to rapidly change the direction of the incoming flow. The interference of the correction force gradually subsides as the flow moves downstream the LE area. Contrary to the impeller SRV2, where large correction forces are exerted near the TE, the contribution of the correction force model is negligible, since there is no change in the camberline curvature causing flow misalignment. The high concentration of turning forces at the LE is evident in all test cases, effectively highlighting the regions of the domain which are more susceptible to numerical stability. This also further accentuates the importance of the correction force model for precise choking predictions. The contours are displayed with a higher aspect ratio for clarity.

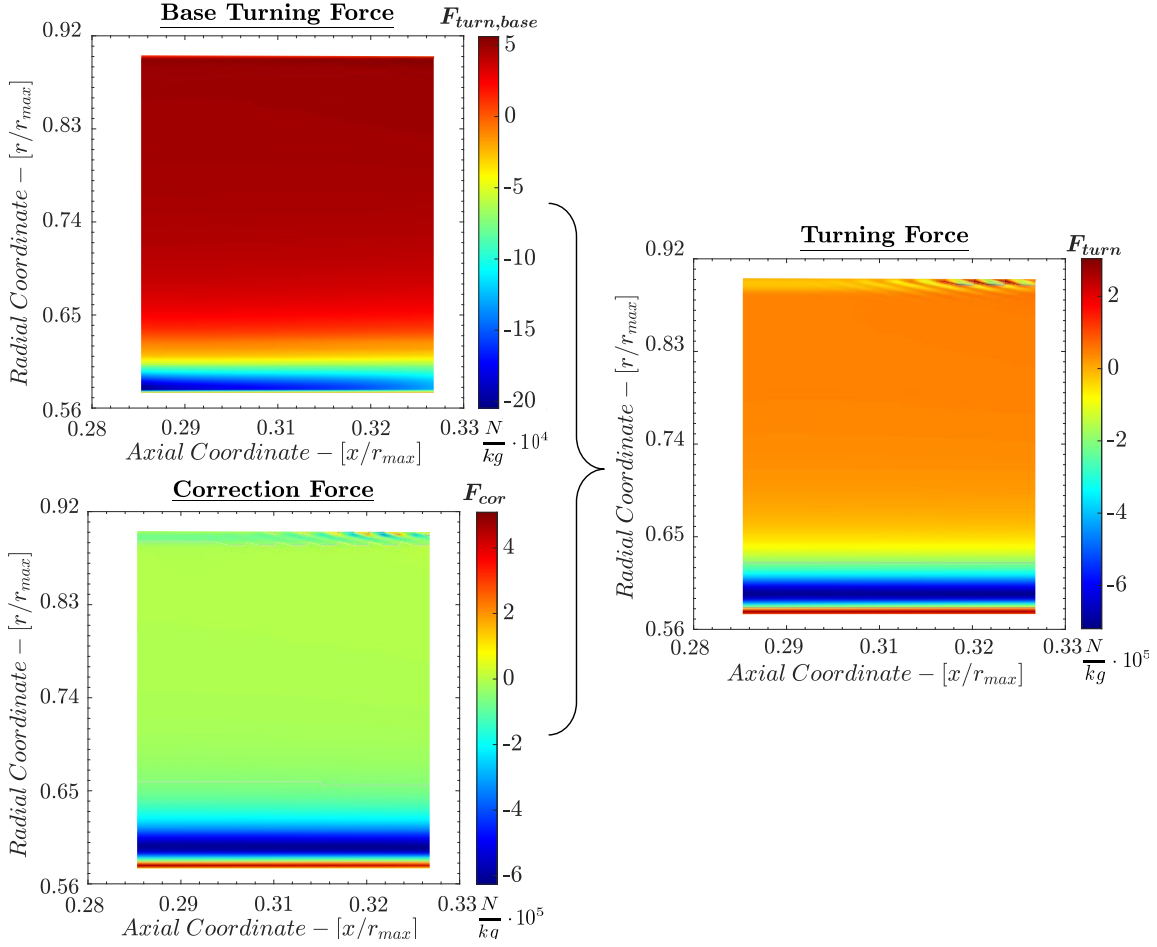


Figure 4.16: Base turning, correction and turning force contours in NASA CC3 vaned diffuser passage.

The applicability of the turning force model to axial compressor blades is tested on NASA rotor 67, a small axial fan rotor. Following a similar approach, the blade, imposed and actual flow angles are plotted along the blade passage at 3 different spanwise locations; hub midspan and shroud (figure(4.17)). Larger misalignments can only be observed near the LE, within the modified curvature region, while downstream the transition region, the misalignment diminishes rapidly. As discussed earlier, the energy input is dependent entirely upon the flow angles of the incoming and outgoing flow. While flow control at the TE is crucial, misalignments in the blade passage, are trivial and do not affect the energy input into the flow, but may affect the choking mass flow, if the flow is substantially misaligned near the throat area. Consequently, errors such as the one at the discontinuous part of the hub (figure(4.17b)), do not affect neither the choking mass flow nor the energy input into the flow. The misalignment at the TE is less than 1% at midspan and the shroud, while the corresponding error at the hub is at the order of 4%. The errors near the throat are less than 3%.

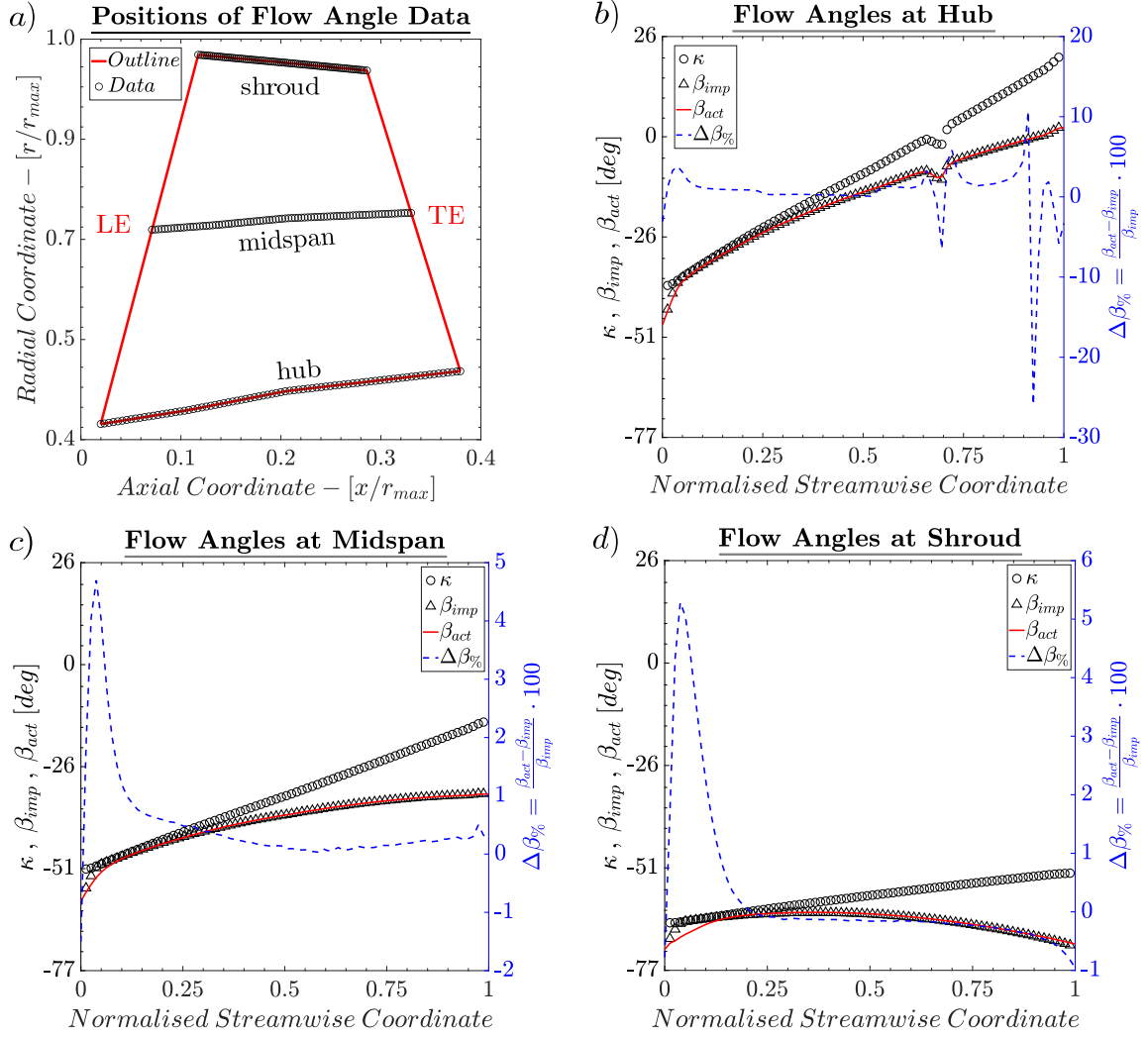


Figure 4.17: Blade, imposed and actual relative flow angle distributions at the hub, midspan and shroud in rotor 67 passage.

The distribution of the turning force components, illustrated in figure(4.18), reveals the presence of large turning forces of increasing intensity from hub to tip as a result of both large base and correction forces at the LE of the blade. The correction force is at the same order of magnitude with the base turning force but reaches substantially higher values, exceeding by almost 60% the magnitude of the base turning force at the LE, near the shroud. This further highlights the need for a characteristic length independent of the radius, since larger correction forces are typically required closer to the shroud. The interference of the correction force diminishes abruptly downstream the LE region, in accordance to the base turning force. The high-turning force region is confined within a very narrow region near the LE, and could potentially trigger numerical instabilities if the local discretisation resolution in the streamwise direction, is not sufficiently dense.

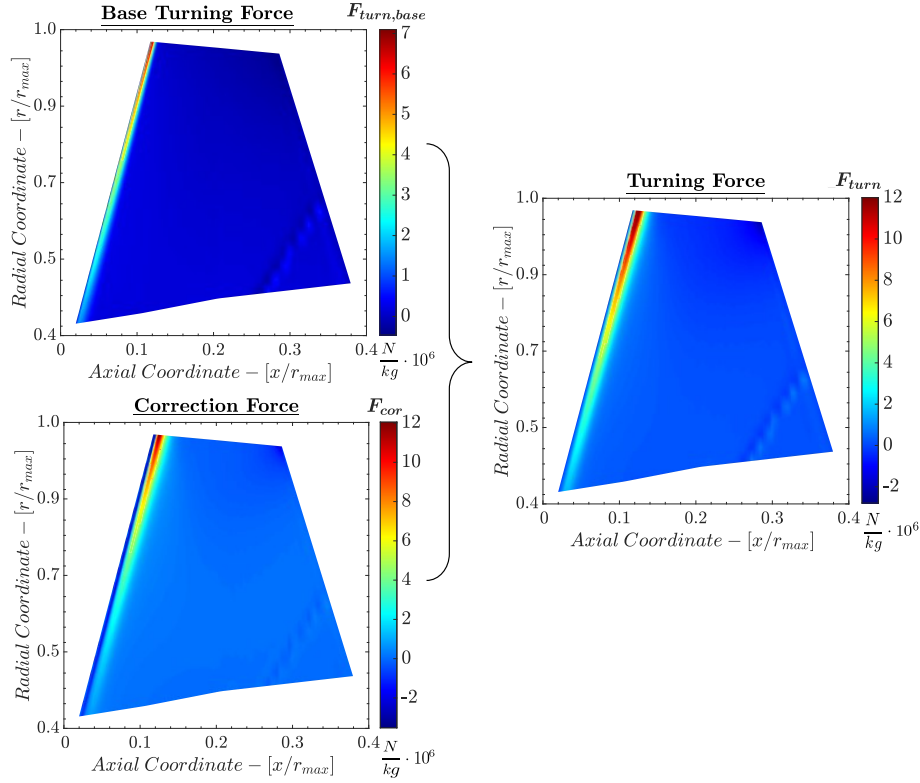


Figure 4.18: Base turning, correction and turning force contours in rotor 67 passage.

The validation of the turning force model is concluded with stator 67, a stator blade located downstream rotor 67. The meridional blade outline along with the positions of the corresponding flow angle data are shown in figure(4.19). The alignment of the actual flow angle with the imposed one is exceptionally well, with the misalignment at the TE being less than 0.5% along all span positions. A noteworthy misalignment can only be seen at approximately 75% of the blade’s meridional length, due to the presence of a small geometric discontinuity in the blade angle distribution. This misalignment is however rapidly alleviated and the flow exits the blade with the imposed flow angle. The turning force contours are omitted since they exhibit similar characteristics to the ones of the rotor; the intervention of the correction force is pronounced within a small area near the LE and rapidly subsides as the flow moves towards the TE.

The limitations of the correction force model are demonstrated, using rotor 37, a geometry representative of an axial compressor rotor. While the inclusion of a correction force model was proven to be exceptionally beneficial, enabling accurate flow turning during transient and steady-state conditions, certain limitations accompany its utilization. The extent of the correction force model’s intervention is adjusted via a two-fold mechanism; it is intrinsically adjusted in the correction force model by

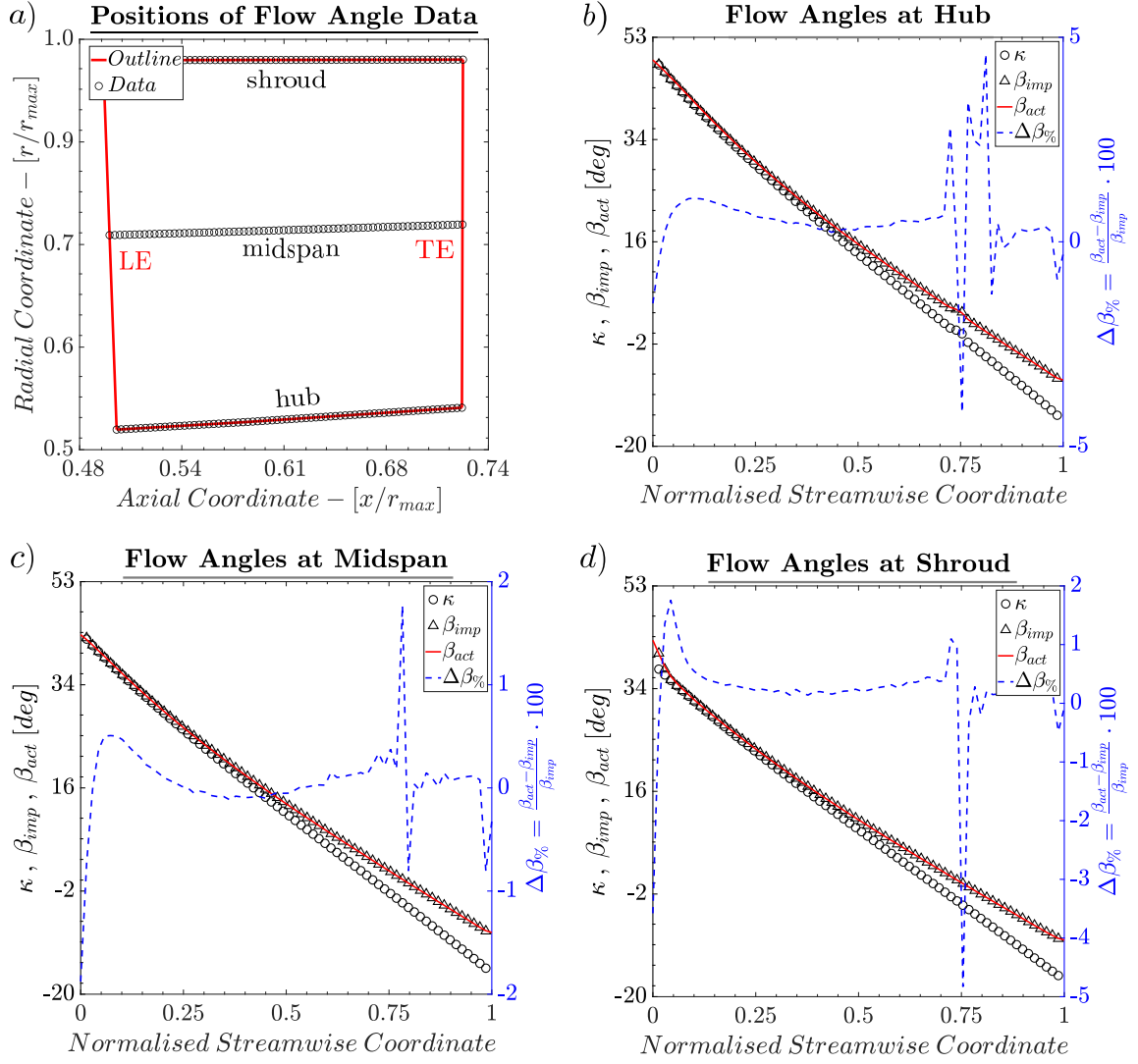


Figure 4.19: Blade, imposed and actual absolute flow angle distributions at the hub, midspan and shroud along stator 67 passage.

determining the local flow misalignment and via a user-defined parameter, K_{cor} . If the latter is set to a value sufficiently high and exceeds a critical value, unrealistic flow behaviour may be triggered, causing local overshoots and undershoots, significantly affecting global aerodynamic performance predictions.

This behaviour is depicted in figure(4.20c,d) with the effect being more pronounced at the shroud, where this unrealistic flow behaviour extends along the entire passage. Higher spanwise positions are generally dominated by larger turning forces and typically, larger correction forces, thus are more prone to overshooting and undershooting. This is evident by examining figure(4.21) which shows the distribution of the turning force components. Moving towards the shroud, the correction force distribution becomes distorted and the phenomenon is exacerbated. It should be

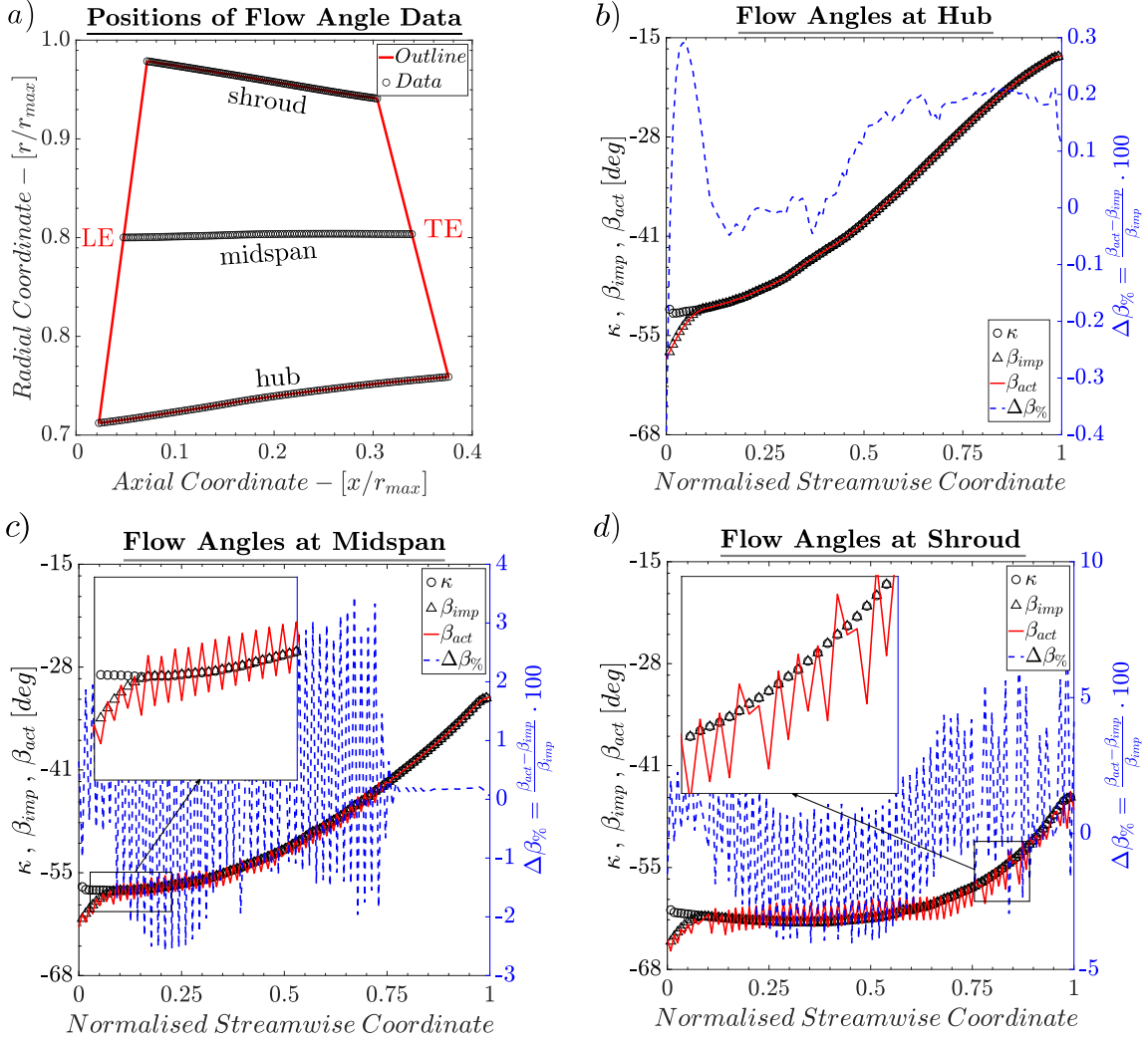


Figure 4.20: Blade, imposed and actual relative flow angle distributions at the hub, midspan and shroud in rotor 37 passage.

noted that the correction force magnitude is one order higher than the base turning force, and is therefore the dominating component. Axial compressor blades usually require smaller correction factors and are more sensitive to this phenomenon, while impellers and vaned diffuser blades generally are more resilient but require higher correction coefficients. For axial compressor blades the correction coefficient is usually set in the range $K_{cor} = [1 - 3] \cdot 10^2$, while for impellers and vaned diffuser blades $K_{cor} = [10 - 20] \cdot 10^2$. The coefficient is constant for all operating conditions and does not require further adjustments.

The universal applicability of the developed turning force model was verified by demonstrating its ability to precisely align the flow with the imposed relative path in passages of different compressor blade geometries. The turning of the flow is isentropic; the turning force is acting perpendicular to the relative flow path. Due

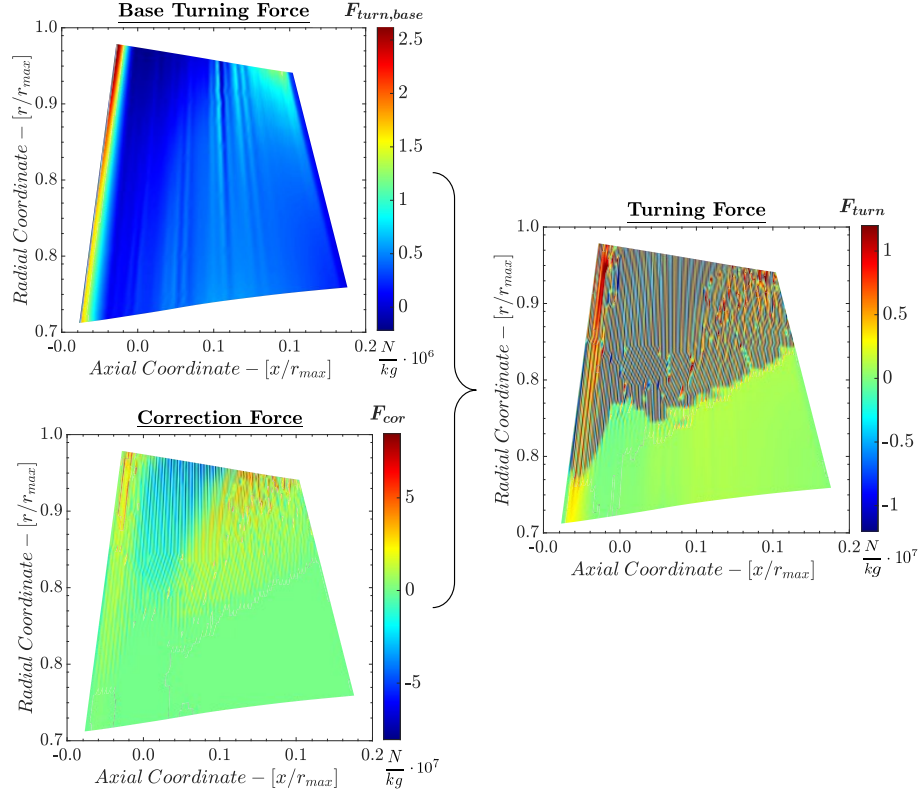


Figure 4.21: Base turning, correction and turning force contours in rotor 37 passage.

to the complex, three-dimensional nature of the relative flow path, an additional verification step was deemed necessary to investigate if entropy generation is generated due to flow turning. Pressure and efficiency characteristics of rotor 37 and impeller SRV2 are obtained at 60% corrected speed (figure(4.22)). Flow turning is introduced via the turning force model, while flow deviation is considered zero. No losses are introduced. Metal blockage is included in the analysis while aerodynamic blockage effects are omitted.

The rotor 37 efficiency characteristics reveal a less than 1.6% efficiency drop in the mid to low mass flow region (4.22a). These are presumably related to blockage effects and to certain limiters that prevent the imposed and actual flow angles during the force decomposition process in the (x, θ, r) axes to exceed a certain value. These will be referred to hereafter as *subsidiary losses* and typically result in efficiency drops of [1% - 2%]. At the higher mass flow range, the higher efficiency drop is related to the formation of shock waves in the passage which are not present at the higher PR settings, causing additional entropy generation. The efficiency characteristic of the impeller SRV2 (4.22b) shows the existence of an entropy generation mechanism in the high mass flow region, which again is associated with the formation of shock waves nearby the impeller TE. In the lower mass flow region, where the flow remains

subsonic, the efficiency is close to 1. It can be concluded, that errors due to non-isentropic flow turning are expected to be within very reasonable limits across the entire operation region of both axial and centrifugal type compressor blades.

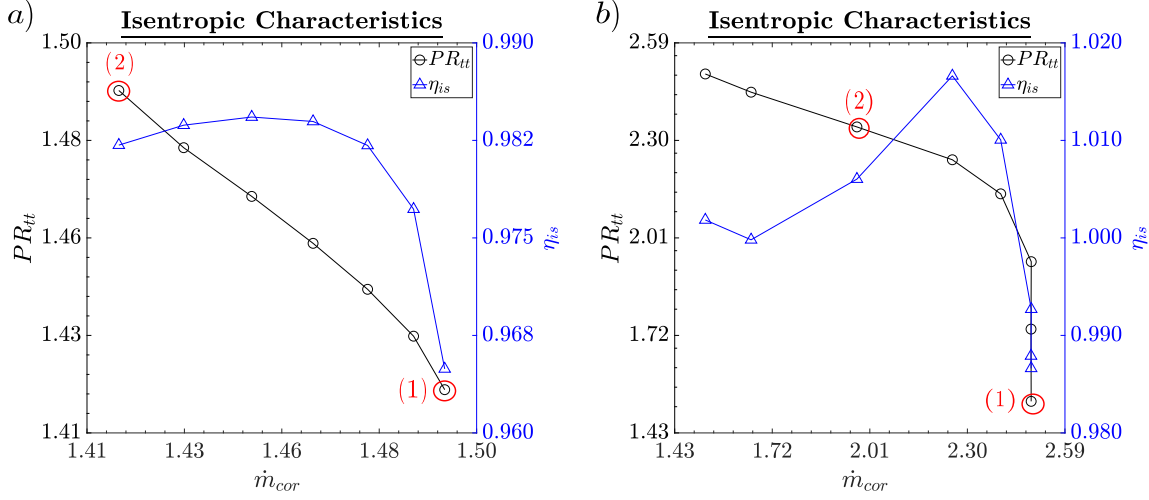


Figure 4.22: Isentropic pressure and efficiency characteristics of a) Rotor 37 and b) impeller SRV2.

4.3 Viscous force model verification

The third constituent component of the body-force model is the viscous force. Due to the absence of shear stresses in the governing equations, energy dissipation and consequently entropy generation relies entirely on the viscous force source term. The viscous force model is a function of the stagnation pressure loss in the relative FoR, a quantity obtained via loss correlation models. While the accuracy of those models is excluded from the verification process since it is not part of present project's scope, the ability of the viscous force model to accurately impose the prescribed amount of losses is essential for global compressor aerodynamic performance predictions. The validity and potential limitations of the viscous force model is performed using two blade geometries; rotor 37 and impeller SRV2. By imposing constant pressure and temperature boundary conditions at the inlet and the outlet, different operating points are obtained by varying the prescribed average pressure loss in the blade passage. Standard ISA atmospheric pressure and temperature are the imposed inlet total pressure and temperature boundary conditions $T_{0,in}$, $P_{0,in}$, while the static pressure at the outlet, P_{out} , is set to a static pressure high enough, to prevent losses due to the formation of shock waves. Blade blockage effects are included in the computational models, while flow deviation is set to zero throughout the entire sweep.

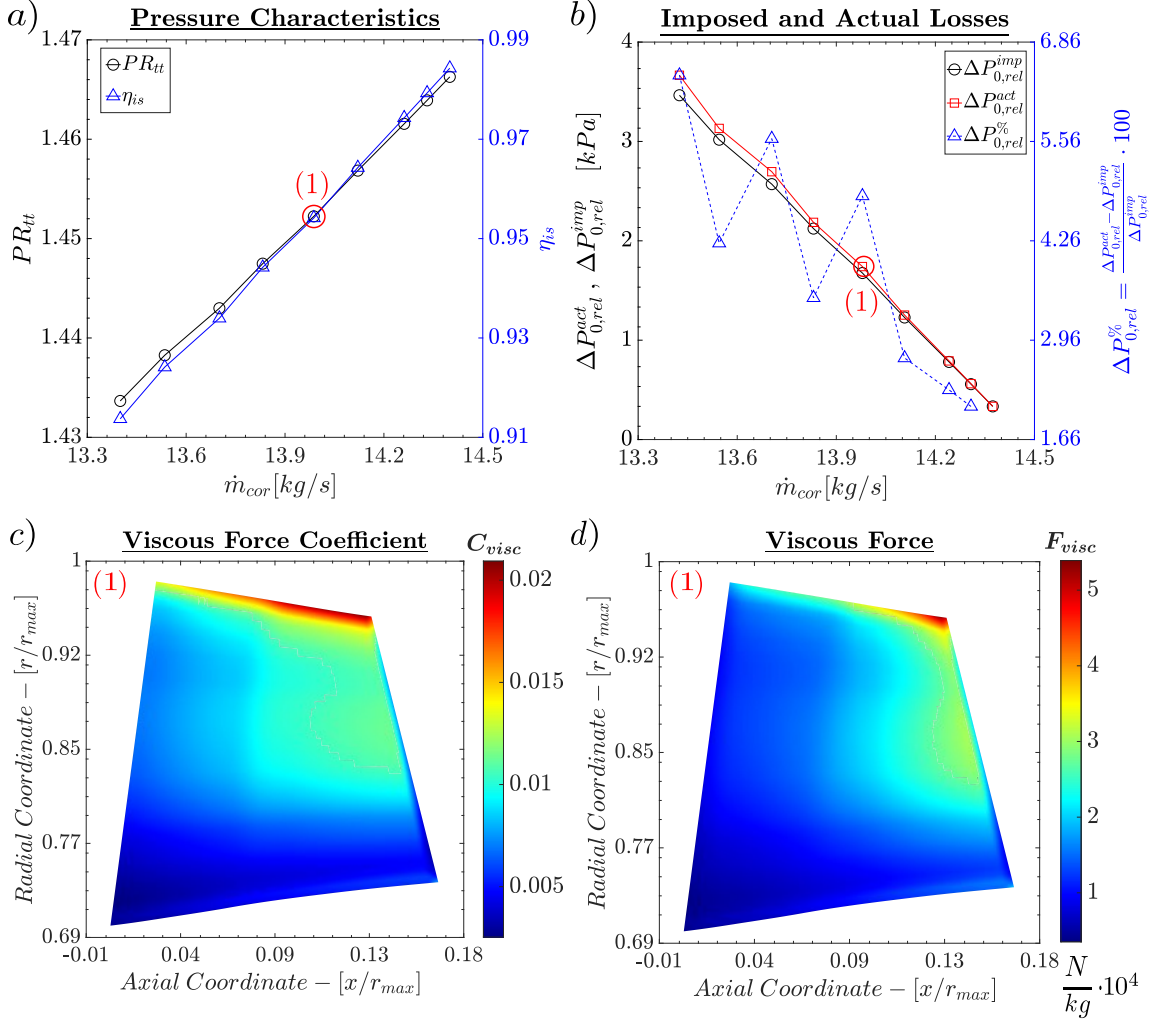


Figure 4.23: Rotor 37 a) pressure and efficiency characteristics at 60% speed obtained by varying the imposed losses, b) comparison of imposed and actual losses, c) viscous force coefficient and d) viscous force at operating point (1).

For the first test case, comprising the computational model of rotor 37, the average imposed pressure loss is in the range $\Delta P_{0,loss} = [250Pa - 3500Pa]$ with an initial increment of 250Pa and subsequent increments of 500Pa. Figure(4.23a) shows the resulting operating points as a function of the mass flow, pressure ratio and efficiency. A distribution coefficient derived from CFD simulations is used to distribute the losses in the blade passage (figure(4.23c)). The measured total pressure loss in the relative FoR is measured at the outlet of the computational domain by subtracting the isentropic, mass flow-averaged stagnation pressure from the resultant one. The prescribed and actual pressure losses in the relative FoR along with the corresponding errors are shown in figure(4.23b). The introduced pressure losses are in excellent agreement with the imposed ones, especially in the lower loss region $[250Pa - 1000Pa]$ where the errors are less than 2.8%. Higher discrepancies are

observed in the higher loss range, with the errors not exceeding 5% except for the highest-loss operating point, where the error exceeded 6.5%. Along the entire sweep, the viscous force model tends to introduce slightly higher losses than the prescribed ones. It is essential to highlight the fact that losses in the range $[250Pa - 1500Pa]$ are a representative range of the losses usually encountered in axial compressor blades at forward flow conditions, thus values higher than $2000Pa$ are rather unrealistic. Another important aspect which should be considered in the analysis of the results, is the imposition of *subsidiary losses* due to blockage and flow turning; these were shown, to a limited extent, to be associated with pressure losses which cannot be dissected in the analysis. Consequently, the viscous force-related errors are expected to be within a very reasonable range of $[0\% - 5\%]$.

Following a similar approach, the validity of the viscous force model is further assessed using the SRV2 impeller blade. A similar sweep to that described earlier is performed and two datasets are obtained; the first is obtained by setting the rotational speed to zero, similar to a stator and the second by setting the rotational speed to 60% of the design speed ($30000rpm$). The geometry of the impeller SRV2 is characterized by large flow path angles and relatively large lean angles, making it an excellent and challenging verification case to evaluate the validity of the viscous force model in a both static and rotating configurations. In both cases, the boundary conditions are kept constant and are sufficiently low to ensure subsonic flow conditions throughout the entire sweep. A uniform distribution coefficient is used to distribute the losses along the blade passage in the stationary configuration (figure(4.24c)) and a CFD-derived coefficient for the rotating impeller simulations (figure(4.25b)).

Figure(4.24a) shows the pressure ratio and efficiency against mass flow of the resulting operating points for the stationary blade passage. Since the impeller is stationary, there is no energy input into the flow and the efficiency is reported in terms of stagnation pressure loss. Figure (4.24b) shows the imposed and actual pressure loss corresponding to each different operating condition. The imposed loss range is $\Delta P_{0,rel} = [0Pa - 10000Pa]$ with an increment of $1000Pa$. The mass flow-averaged pressure loss imposed by the viscous force model is in good agreement with the prescribed one along the entire sweep. The errors are within the range $[6.3\% - 6.8\%]$ with the discrepancies becoming slightly larger in the higher pressure loss range, which is not representative of realistic pressure losses in impellers. It should be highlighted once again, that the entanglement of subsidiary, ostensibly isentropic modelling aspects, i.e. flow turning and blockage effects in the overall entropy generation along the duct is expected to exacerbate the errors shown in figure(4.24b). Figures (4.24c,d)

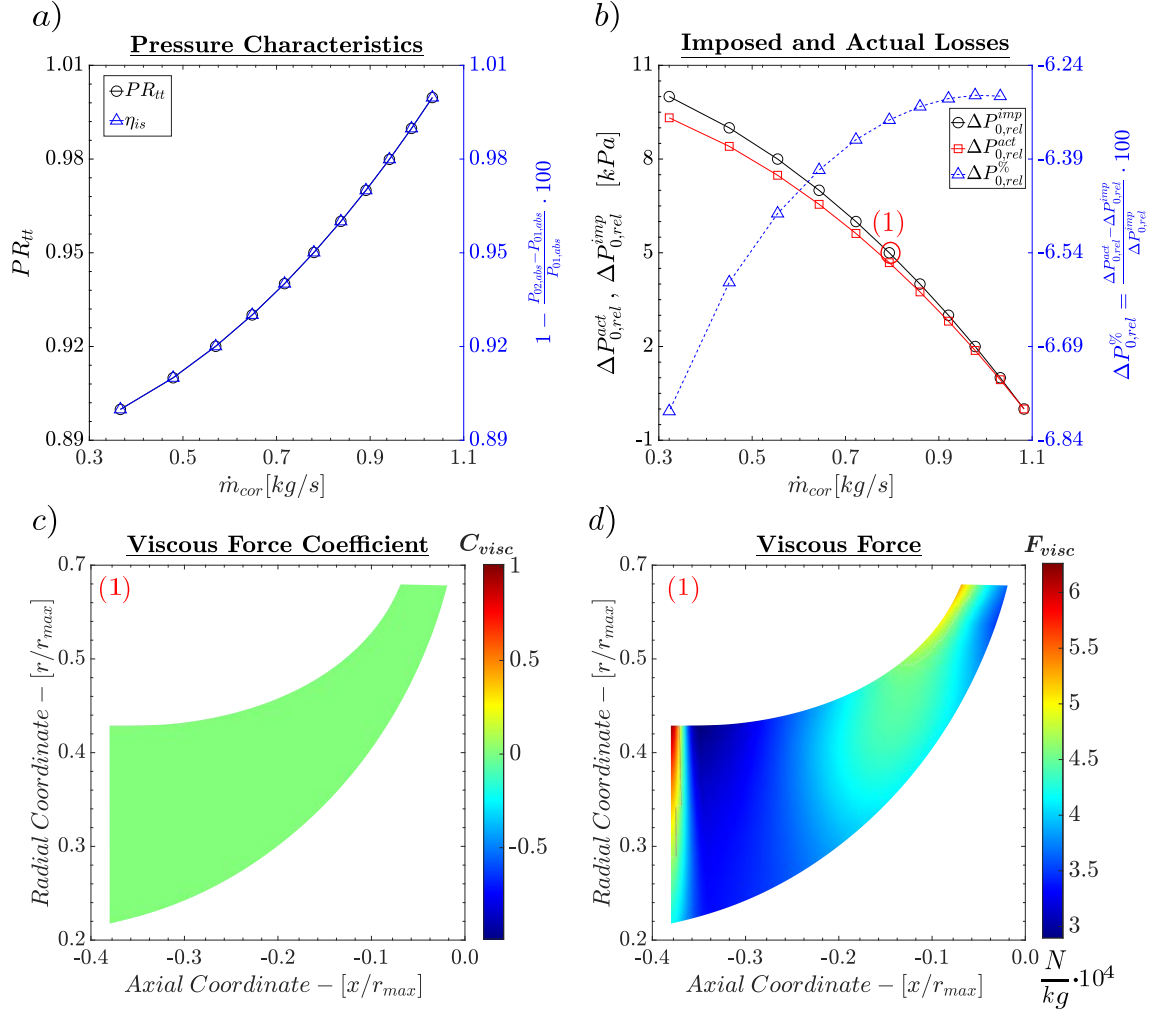


Figure 4.24: Impeller SRV2 modelled as stator a) pressure and efficiency characteristics obtained by varying the imposed losses, b) comparison of imposed and actual losses, c) viscous force coefficient and d) viscous force at point at point (1).

illustrate the viscous force coefficient and the corresponding viscous force distribution along the duct. Despite the entirely uniform distribution coefficient imposed in the first sweep, the viscous force distribution is not uniform; this stems from the fact that the viscous force coefficient is applied to the pressure loss, i.e. $\partial P_{0,rel}/\partial m$ term and not the viscous force itself. Since the viscous force is a function of the geometric flow path angle as well as other flow-field variables, it will inevitably vary in accordance to the local flow conditions of each cell. The forces increase progressively in the streamwise direction, as the flow moves towards the TE, due to increase in the flow path angle and changes in the flow-field variables. Highest dissipation is imposed at the LE, in the modified curvature region, due to the abrupt change in the flow direction and at the shroud, near the TE. This provides a natural viscous force distribution mechanism, which nevertheless is incapable of reproducing complex

viscous phenomena such as tip clearance losses, tip leakage vortices, flow separation, boundary layers etc. which are captured by CFD-derived coefficients.

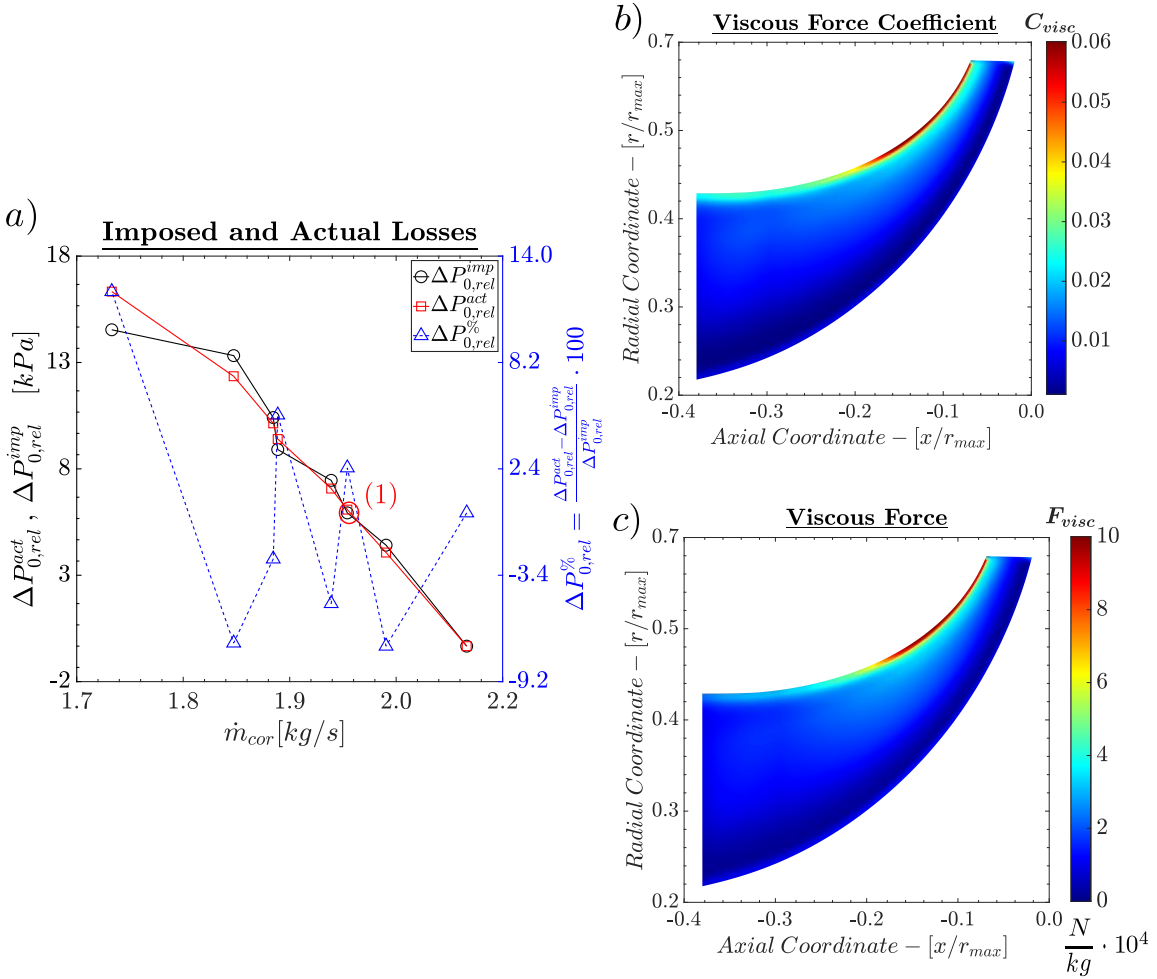


Figure 4.25: Impeller SRV2 rotating at 60% NRT a) comparison of imposed and actual losses, c) viscous force coefficient and d) viscous force at point at point (1).

An identical computational domain is subsequently used to perform the last validation step. To further verify the validity of the viscous force model in rotating domains as well as the distribution coefficient concept, the viscous force is distributed according to a CFD-derived coefficient (figure (4.25b)). Due to the significant change in the mean streamline radius, the applied viscous force is adjusted by the correction factor proposed by Aungier (eq.(3.160)) [68]. The corrected prescribed pressure loss (figure (4.25a)) is then compared against the mass-flow averaged pressure loss at the outlet of the compressor. The actual pressure loss data correlates well with the imposed ones, except for the last two, highest-loss operating points where the errors are approximately -8.2% and 12.5% respectively. The prescribed losses in the highest range are excessively high and are not representative of realistic losses in an impeller

passage at forward flow operation. The recorded discrepancies in the high loss range are consequently not expected to compromise forward flow predictions. The accuracy of the model across the rest of the lower pressure loss range is very satisfactory and is within the same error range to that of rotor 37.

4.4 Distribution coefficient verification

Distribution coefficients constitute an integral numerical and modelling aspect of both the turning and the viscous force model. The definition of two distribution coefficients is prerequisite for the complete definition of the body force model; the viscous force and deviation angle distribution coefficients. These coefficients are dictating the distribution of the local pressure loss and deviation at a finite volume scale. The foundation of their numerical definition ensures that given any averaged value, e.g. average pressure loss or deviation along a passage, the averaged quantity will be the same regardless of the applied distribution pattern. In practice, this would imply that given a prescribed average pressure loss or deviation, the averaged value at the outlet of the passage should not change regardless of the applied distribution coefficient. This would consequently imply that global aerodynamic performance predictions should not be dependent upon the coefficients unless they induce phenomena which potentially could compromise control of the flow, e.g. flow separation due to concentration of excessively large losses.

To verify the validity of this hypothesis as well as the validity from a mathematical and numerical implementation perspective, two test cases are performed using rotor 37 geometry. In the first test case, rotor 37 is modelled as an isentropic compressor (no viscous forces) with metal blockage. Different operating points are obtained by varying the deviation angle. In the second test case, different operating points are obtained by varying the imposed pressure losses, while the deviation angle is set to zero. In both test scenarios, the boundary conditions are identical and invariable throughout the entire sweep. Each sweep is performed twice; initially a uniform coefficient is applied and subsequently a CFD-derived one.

The operating points of the deviation angle sweep are given as a function of the pressure ratio, efficiency and the mass-flow in figure(4.26). Both the pressure ratio and the efficiency characteristics are in excellent agreement, with the main difference being the different position of the operating points along the characteristics. This indicates that whenever the distribution of the quantities under consideration changes, slightly different boundary conditions are required to exactly match operating points,

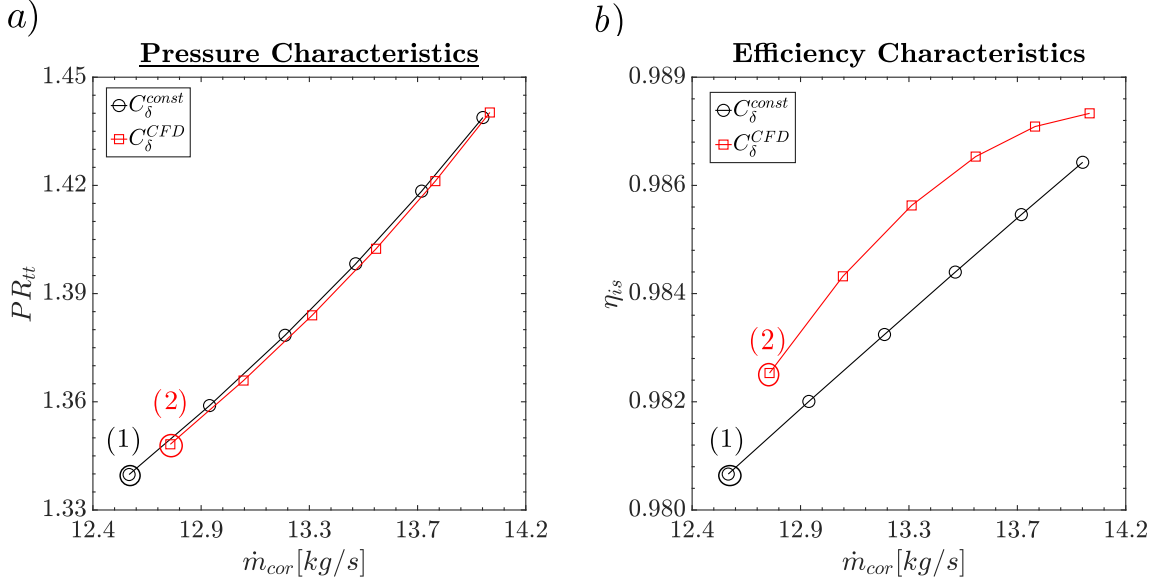


Figure 4.26: Rotor 37 a) pressure and b) efficiency characteristics obtained by varying the imposed deviation angle using uniform and CFD deviation coefficients and no losses.

nevertheless the operating points are expected to move along the same characteristic. The results clearly confirm the validity of the distribution coefficient method but also confirm the hypothesis made earlier; global compressor aerodynamic performance is dependent upon the averaged flow-field conditions and not on the distribution of the flow-field quantities along the passage.

Figure (4.27) shows the spanwise distribution of the pressure and temperature ratio at the TE along with contours of the imposed deviation angles in the passage. The effect of the distribution coefficient on the spanwise profiles is evident, substantially altering the spanwise profiles, while preserving the averaged deviation angle. The CFD-derived coefficient introduces the majority of the flow-deviation in the vicinity of the TE, unlike the uniform coefficient which is introducing it gradually. The CFD coefficient is imposing higher flow deviation in the region between midspan and the hub and lower deviation in the region between midspan and the shroud (figure(4.27d)). Compared to the uniform coefficient case, this results in reduced flow turning and energy input near the hub but increased flow turning and consequently energy addition near the shroud. The aforementioned spanwise variation in the flow turning distribution is reflected on both the spanwise stagnation pressure and temperature ratio profiles downstream the TE, shown in figure(4.27a,b). Since only flow turning is imposed (no viscous force), both profiles exhibit similar behaviours. The flatter profile of the pressure ratio graph near the shroud could presumably be associated with subsidiary losses due to turning or blockage-induced losses, potentially

indicating their location.

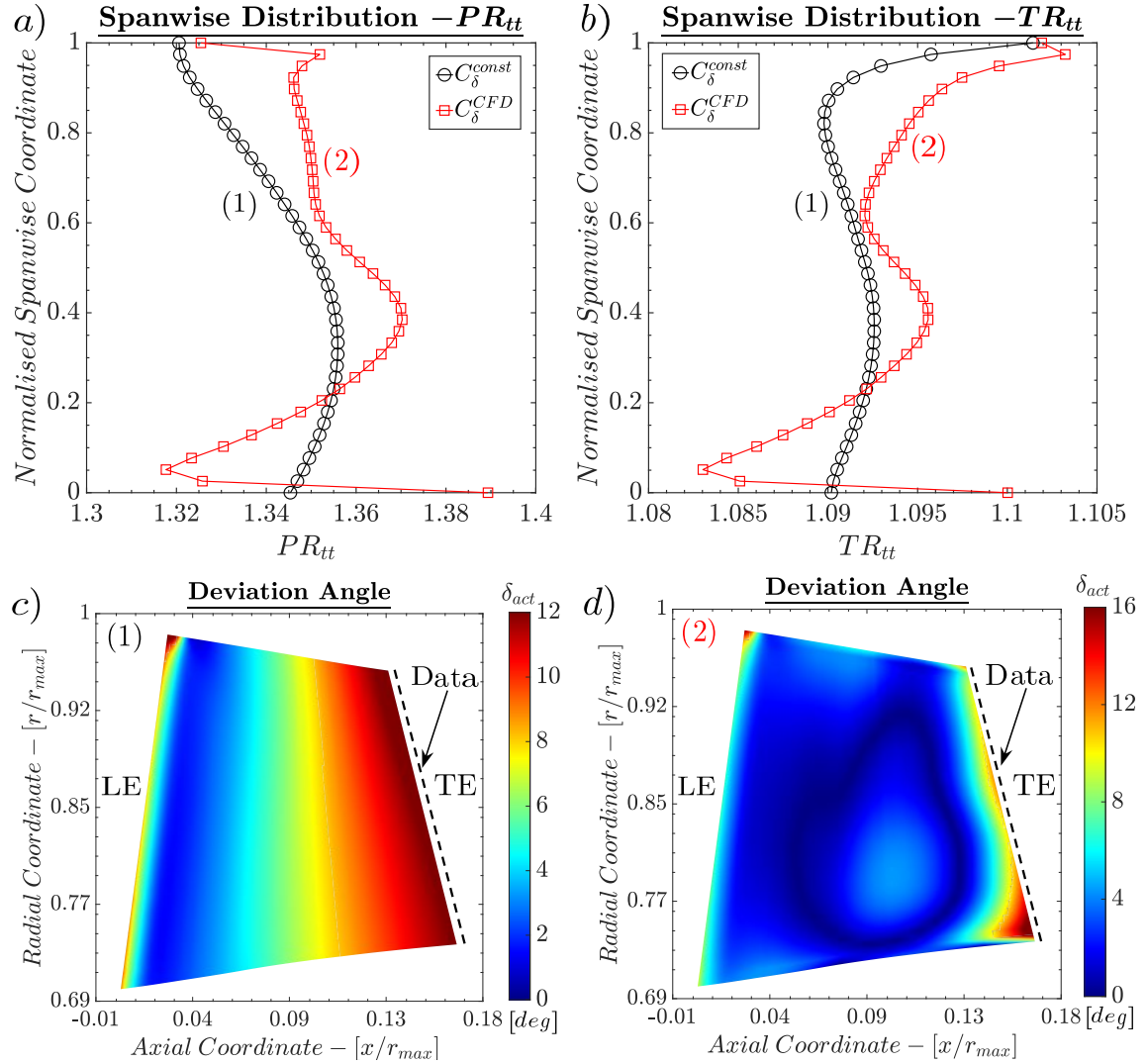


Figure 4.27: a) Total pressure and b) total temperature ratio profile at the TE of rotor 37 obtained using uniform and CFD viscous force coefficients. The distributions of the deviation angle corresponding to operating points (1) and (2) are shown in c) and d).

The validity of the distribution coefficient concept is further investigated by comparing the mass-flow-averaged relative flow angles downstream the TE against the imposed average deviation angles. Figure(4.28) shows the imposed relative flow angle (filled circles) along with the measured, mass-flow-averaged, flow angles just downstream the TE obtained using both uniform and CFD coefficients. The corresponding errors are also included in the same graph. The averaged flow angles using both coefficients correlate exceptionally well with the imposed angles along the entire sweep. The recorded errors for both sweeps do not exceed 2.61%, with the CFD-coefficient curve (red) being consistently more accurate than the uniform one (empty circles) achieving errors less than 1.4%.

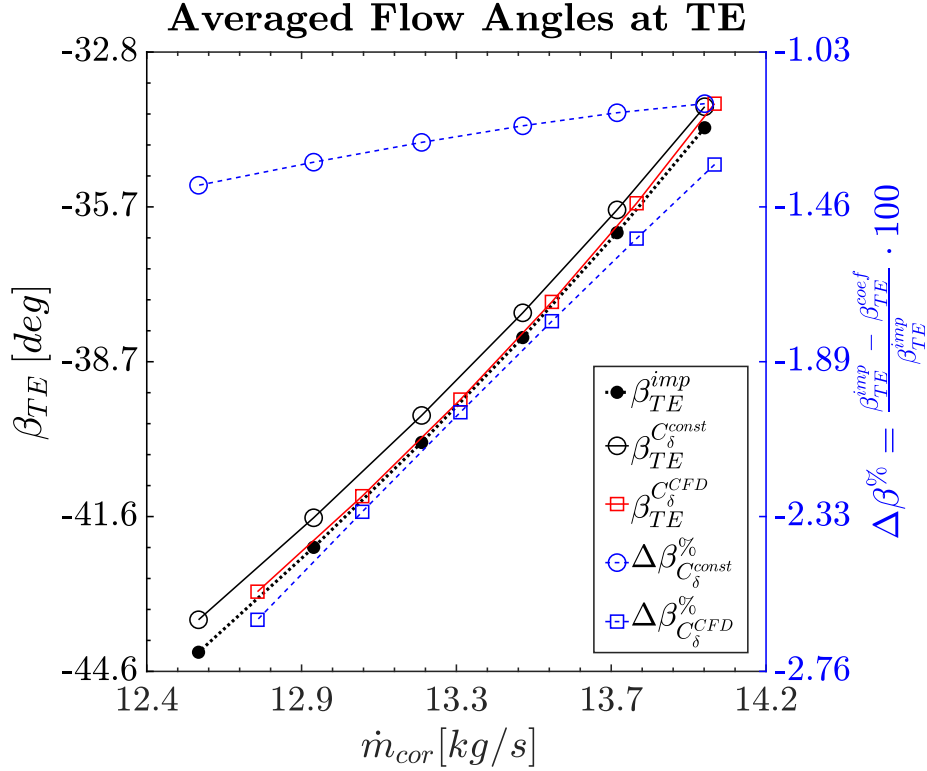


Figure 4.28: Comparison of imposed and measured flow angles at the TE using different deviation coefficients.

A similar approach is followed to verify the validity of the viscous force distribution coefficient. Even though its validity has partially been investigated in section(4.2), a similar verification process is implemented for the sake of completeness. A sweep consisting of 5 operating points is carried-out using an identical computational model of rotor 37 and a similar approach; the boundary conditions are constant, however the deviation angle is set to zero, with changes in the operating points resulting due to the imposition of different pressure losses. The operating points are obtained by applying both a uniform and a CFD viscous force distribution coefficient, thereby recasting the distribution of the imposed averaged total pressure loss along the blade passage. The pressure and efficiency characteristics derived using the two coefficients are shown in figure(4.29). The pressure and efficiency characteristics exhibit identical trends, with the main differences being the slightly underestimated pressure ratios and the slightly higher efficiencies of the uniform coefficient model which in both cases does not exceed 0.9%. The offset between the two curves could potentially indicate that the uniform coefficient model introduces slightly higher deviation and less losses compared to the CFD one, but the discrepancies could also partially originate to changes in the mass flow predictions, shifting the characteristics along the axial axis. Similar to the previous test case, there is a slight shift of the operating points when

the boundary conditions are the same but different coefficients are used.

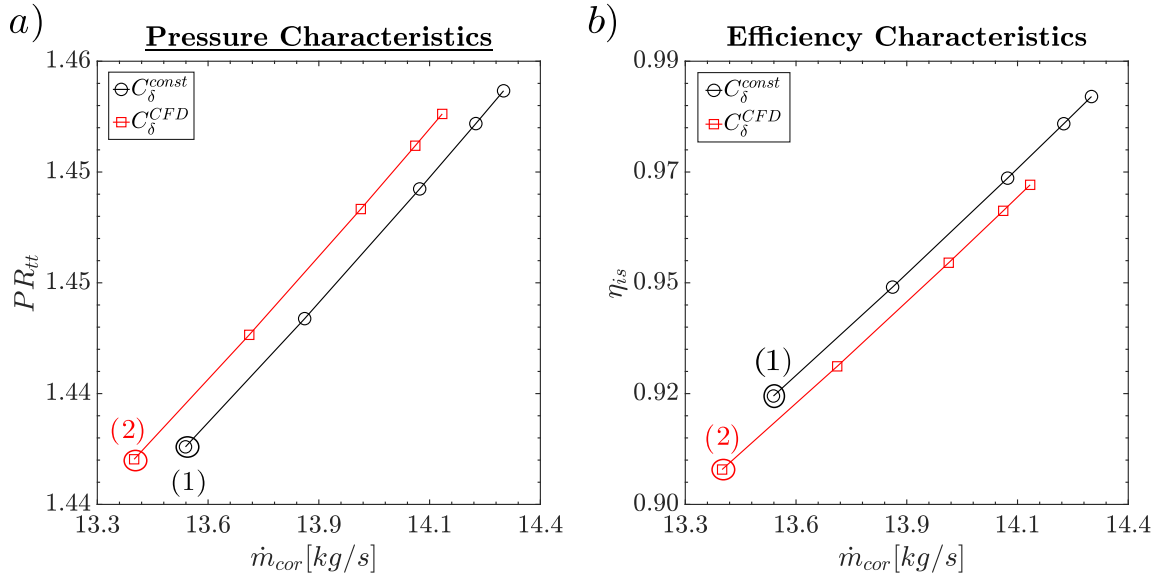


Figure 4.29: Rotor 37 a) pressure and b) efficiency characteristics obtained by varying the imposed total pressure loss and applying a zero deviation angle using uniform and CFD deviation coefficients.

The transformation of the pressure and temperature ratio spanwise profiles due to the change in the distribution of the imposed averaged pressure loss is shown in figure(4.30a,b). The displayed profiles correspond to the highest pressure loss setting and consequently the lowest pressure ratio and mass flow operating points. Even though the pressure ratio difference between the two operating points is negligible, their position along the efficiency characteristic is different. The pressure ratio characteristic is shifted downwards, while the corresponding efficiency characteristic is shifted slightly upwards. This indicates that less dissipation is introduced by the constant coefficient model. This is evident by the temperature ratio spanwise profiles shown in figure(4.30). The CFD distribution coefficient reconstructs the viscous force-field in a more realistic way, reproducing viscous phenomena such as the mixing-out of the tip leakage vortex at the shroud, locally forming a high-loss region (figure(4.30d)). The presence of that high-loss region requires additional energy input, imparted by the turning force model, to achieve the same flow turning. This additional energy input manifests in the form of a rise in the stagnation temperature ratio in the vicinity of the TE (figure(4.30b)). Despite the large energy input in the shroud region, the increase in the pressure ratio diminishes remarkably due to the presence of large viscous forces, thus the achieved pressure ratio is approximately the same but the efficiency lower.

To conclude the verification of the viscous force coefficient, mass flow-averaged

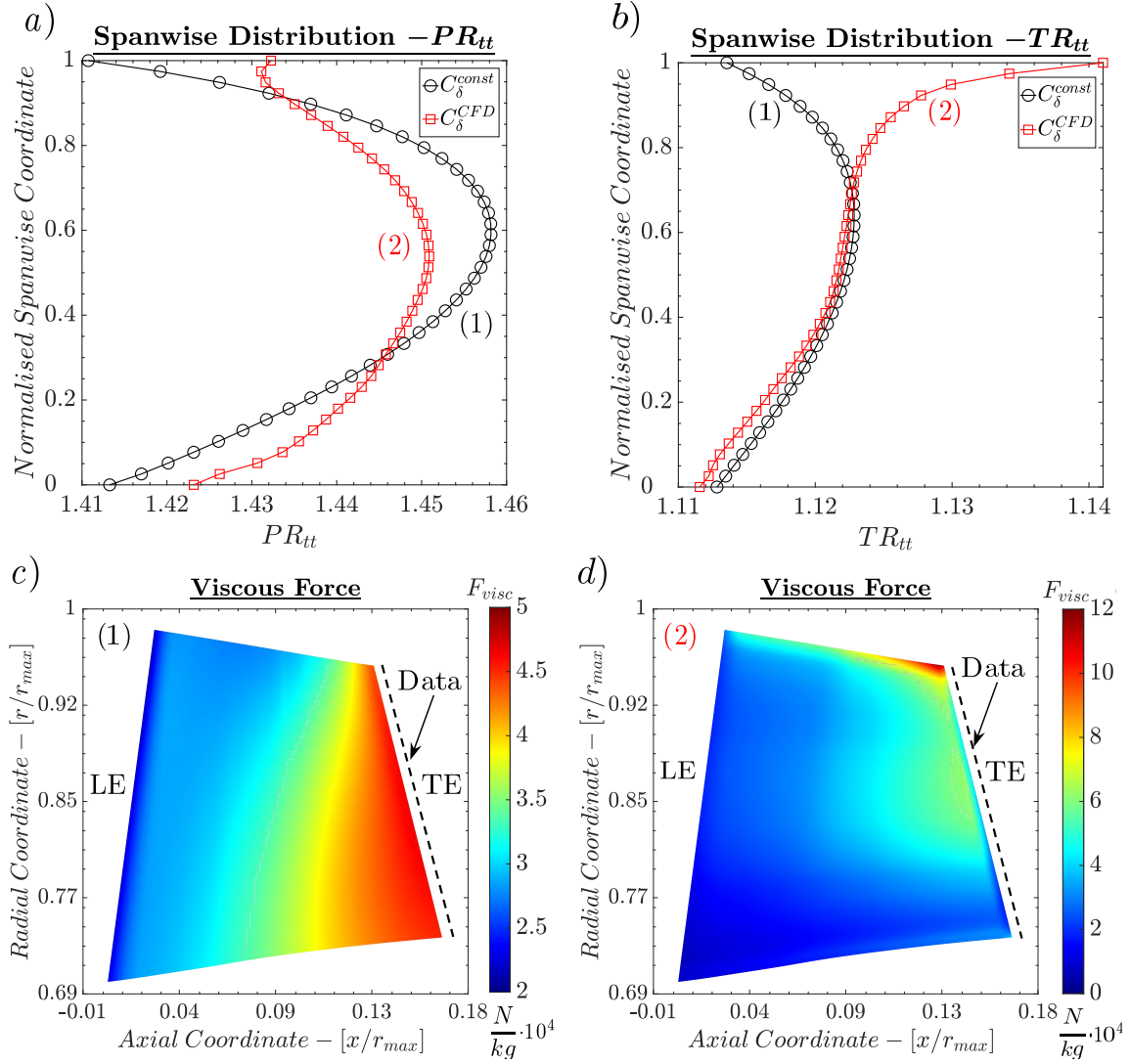


Figure 4.30: a) Total pressure and b) total temperature ratio profile at the TE of rotor 37 obtained using uniform and CFD viscous force coefficients. The distributions of the viscous force corresponding to operating points (1) and (2) are shown in c) and d).

stagnation pressure losses are measured at the outlet of the domain. To limit the interference of subsidiary losses, a correction is applied to all the measured pressure losses by removing the amount of subsidiary losses measured in a fully isentropic case. Figure(4.31) shows the imposed and measured pressure losses using the two aforementioned coefficients for the viscous force along with the corresponding errors. The errors are reported with respect to the imposed values. The measured losses correlate remarkably well with the imposed losses, with the errors corresponding to the CFD and uniform coefficient curves not exceeding 1% and 4% respectively. The hypothesis regarding the imposition of slightly higher losses by the CFD coefficient model is not directly evident by the measured pressure losses, as the uniform coefficient model is ostensibly introducing slightly higher losses. The lower efficiencies of the

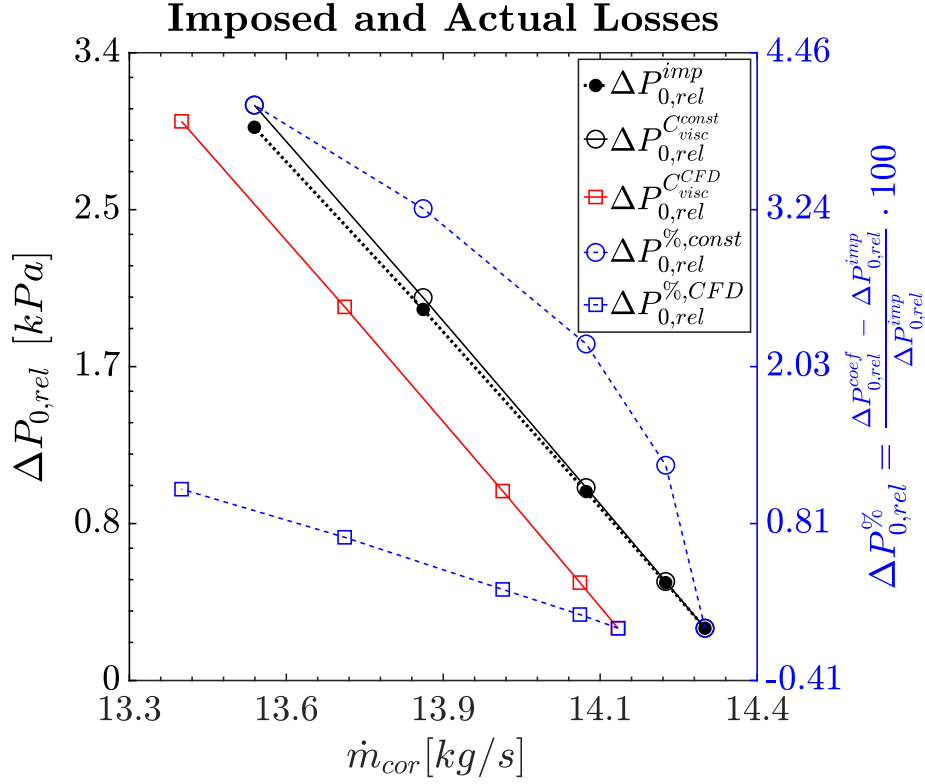


Figure 4.31: Comparison of imposed and measured pressure losses at the TE using different viscous force coefficients.

CFD distribution model and the offset between the pressure and the efficiency curves of the two models are stemming from higher subsidiary losses introduced by the CFD model. Their effect is removed from the data in figure(4.31), but is captured by the pressure ratio and efficiency characteristics(4.29). The implications of subsidiary losses on global aerodynamic performance was investigated in section(4.2)

The verification of the distribution coefficient concept and its implications on global aerodynamic performance are finally evaluated by obtaining the 60% speed characteristics of two different compressor geometries, rotor 37 and the CSTAR centrifugal compressor with a vaned diffuser. The characteristics corresponding to rotor 37 are obtained using combinations of uniform and CFD-derived coefficients for both the deviation and the viscous force. The characteristics of the latter are derived by imposing CFD coefficients only to the viscous force. The spanwise distribution profiles of the pressure, temperature ratio and efficiencies are compared against CFD profiles at 3 operating conditions (near choke, off-design, near stall) to evaluate the ability of the coefficients to generate more realistic spanwise profiles. A single CFD distribution coefficient, derived at near-choke conditions has been used for all operating points.

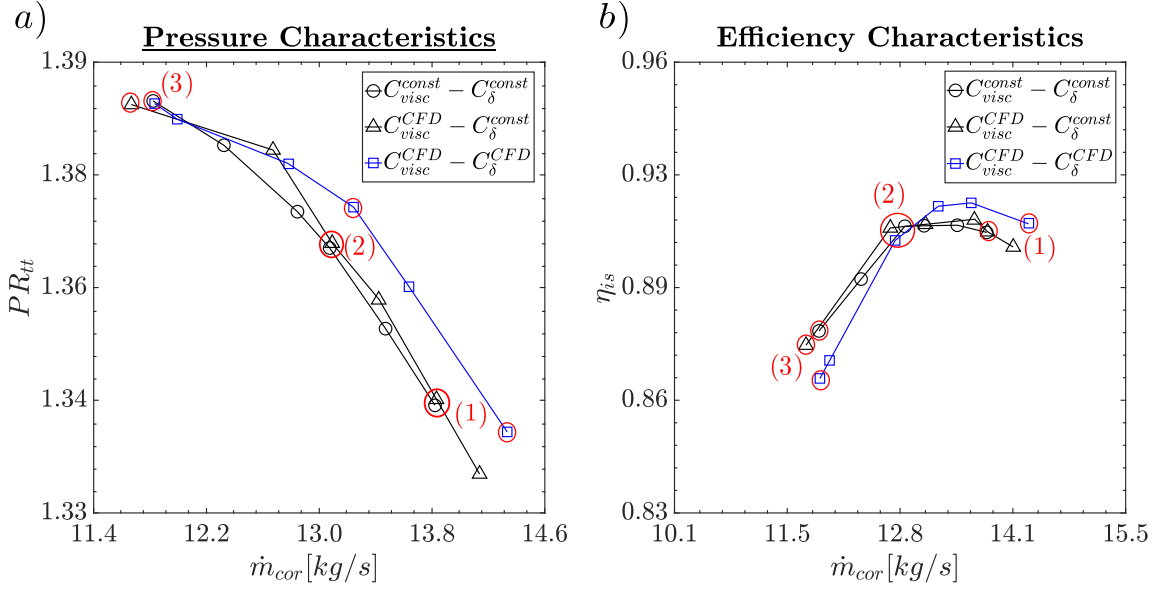


Figure 4.32: Pressure ratio and efficiency characteristics of rotor 37 obtained using different combinations of deviation and viscous force distribution coefficients at 60% design speed.

Figure(4.32) shows the pressure and efficiency characteristics of rotor 37 obtained using different combinations of uniform and CFD coefficients. The baseline model (circles) incorporates only uniform coefficients, the second (triangles) comprises a CFD viscous force coefficient and a uniform deviation coefficient, while the last one comprises entirely CFD coefficients. The average deviation and pressure losses at individual operating points are not predetermined but transiently estimated by the empirical loss and deviation models based on the instantaneous state of the flow-field at the passage inlet and outlet. Both the pressure and the efficiency characteristics obtained using all 3 combinations of distribution coefficients are in very good agreement with each other. In the higher mass flow region, the curve derived using CFD coefficients only, slightly overpredicts the pressure ratio while slightly underpredicting the efficiency, indicating increased energy addition due to flow turning and higher losses. The efficiency in the lower mass flow region exhibits a reverse trend, estimating higher efficiencies but similar pressure rise. The baseline and the second model (circles and triangles) are in excellent agreement with each other, displaying minor, inconsequential differences. Considering the analysis performed in the previous sections, it is reasonable to deduce that the variations in the aerodynamic performance predictions are linked to changes in subsidiary loss sources, the intensity of which varies between operating points, as well as slight differences in deviation and loss predictions obtained by the empirical models.

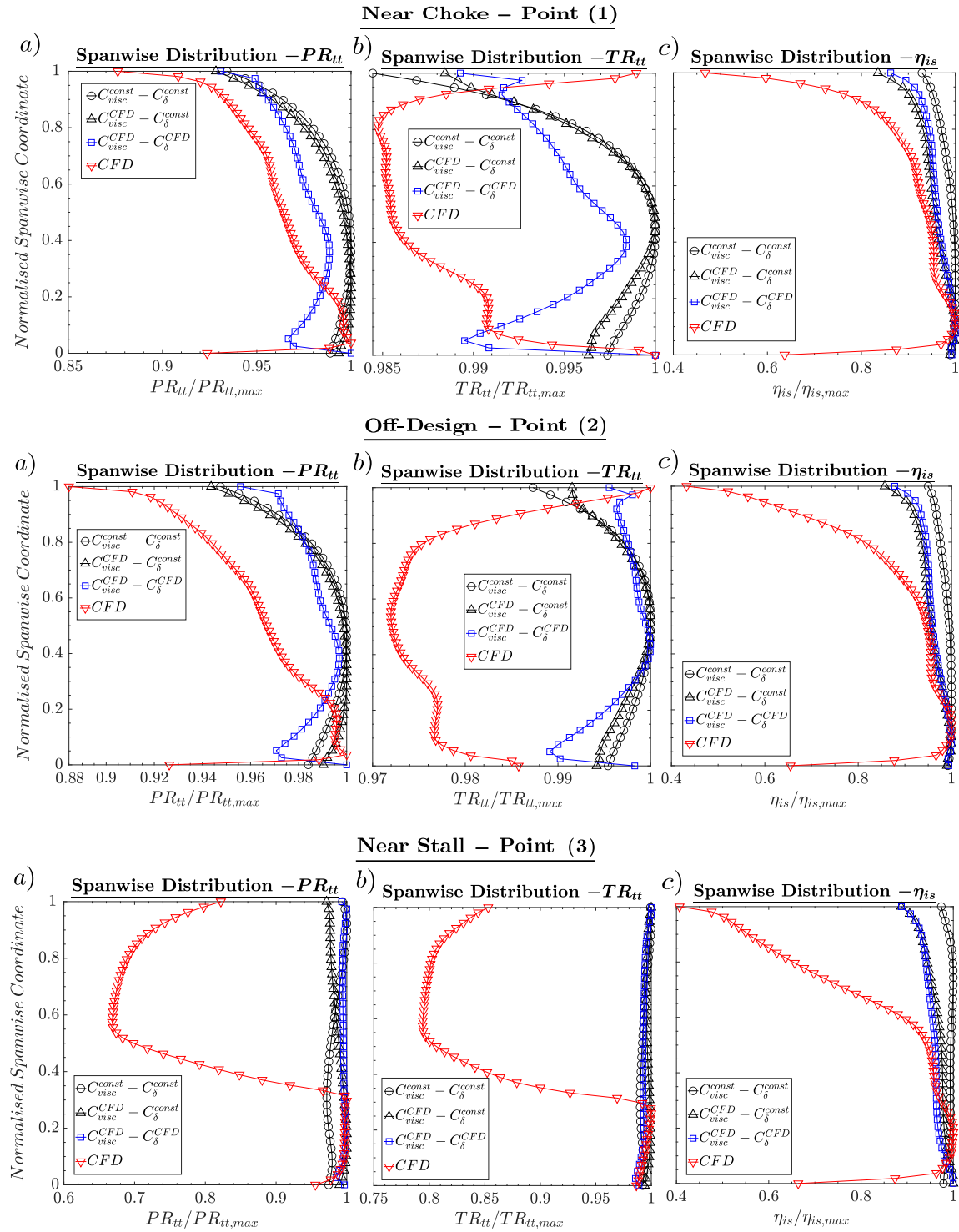


Figure 4.33: Spanwise profiles of pressure ratio, temperature ratio and efficiency of rotor 37 at near choke, off-design and near stall operating conditions.

To investigate the accuracy and potential benefits of the distribution coefficient method, spanwise pressure ratio, temperature ratio and efficiency profiles corresponding to all three models were obtained and compared against profiles obtained from a CFD model. Since the characteristics obtained with the code were found to

overpredict pressure ratio and underpredict efficiencies, the data were normalised with the corresponding maximum values. Data are obtained at near choke, off-design and near-stall operating conditions. The spanwise profiles of the aforementioned quantities are illustrated in figure(4.33). At near-choke conditions the pressure ratio and efficiency characteristics are in good agreement with the CFD ones, with the most notable differentiations being concentrated near the hub and the shroud. These originate from viscous phenomena such as boundary layer at the hub and tip leakage vortices at the shroud which precipitate substantial decrease in the pressure rise and the corresponding efficiencies as well as increase in the required local energy input to achieve turning of the flow (figure(4.33(1b) - red curve). The pressure ratio and efficiency profiles correlate reasonably well with the CFD data, while the highest discrepancies between the code and the CFD data are concentrated primarily near the hub and the shroud. The temperature ratio profile of the third model (CFD coefficients) is the most accurate, predicting the increased temperature ratio distribution near the hub but is unable to reproduce the large total temperature rise near the shroud, distributing the energy input more uniformly along the blade. It should be noted that the scale of the x-axis showing the temperature ratios is representative of changes in the order of just 1.5%. Both the pressure ratio and the efficiency profiles correlate well with the CFD data, with the largest discrepancies being more pronounced near the hub and the shroud region where the intensity of the viscous phenomena is exacerbated. The underestimation of the dissipation intensity near the hub and the shroud is also reported by Kottapalli [85] and Brand [88] and is partially related to the limitations of their viscous force extraction method which was also adopted in the present work.

As the operating point moves towards off-design conditions, the profiles become less accurate, with the regions near the hub and the shroud being the least accurate ones. At near-stall conditions, the distribution of the forces and consequently of the corresponding profiles changes dramatically, due to flow separation near the shroud which significantly augments losses, resulting in low energy input and lower pressure rise and efficiency in the region between midspan and the shroud (figure(4.33(3)) - red curves). Since the applied distribution coefficients were derived at near-choke conditions, they are incapable of reproducing such flow conditions. The noteworthy variations in the spanwise profiles at different operating points on the characteristic, indicate that the inclusion of coefficients representative of more operating points is necessary to improve accuracy in axial compressor blades.

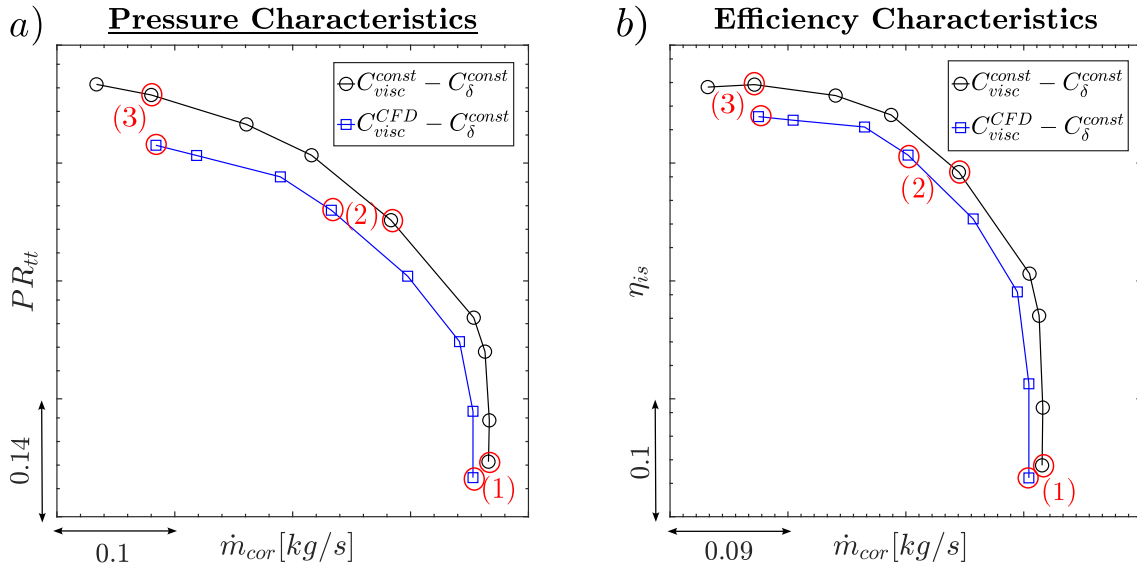


Figure 4.34: Pressure ratio and efficiency characteristics of CSTAR centrifugal compressor obtained using different viscous force distribution coefficients.

The assessment of the distribution coefficient concept is concluded following a similar approach in a centrifugal compressor with a vaned diffuser. Two models are used; the first incorporates uniform coefficients only, while the latter incorporates a CFD coefficient for the distribution of the viscous force and a uniform for the distribution of the deviation. The pressure and efficiency characteristics are shown in figure(4.34). Both the pressure rise and efficiency predictions of the second model are consistently lower compared to the baseline model by approximately 5% and 3% respectively. The changes stem from the a combination of sources mentioned earlier, i.e. subsidiary losses, different predictions of the empirical models and small errors related to the viscous force and its distribution (see figure(4.31) and section (4.3)).

Spanwise profiles of temperature ratio, pressure ratio and efficiency are obtained using the through-flow code and the 2 models described earlier. These are subsequently compared against RANS simulations of the same compressor geometry at three operating points to assess their capabilities and limitations; near-choke, off-design and near-stall conditions (figure(4.35)). It is evident that the distribution profiles of all the plotted quantities do not undergo any noteworthy changes regardless of the operating state of the compressor. The beneficial impact of the CFD-derived viscous force coefficient compared to the uniform one is particularly evident in the temperature ratio and efficiency distributions. The profiles are in reasonable agreement with the CFD results, providing substantial benefits compared to the flat, linear profile generated by the baseline configuration. Similar to rotor 37, the limitations of the coefficients are more pronounced near the hub and the shroud region, where the imposed losses

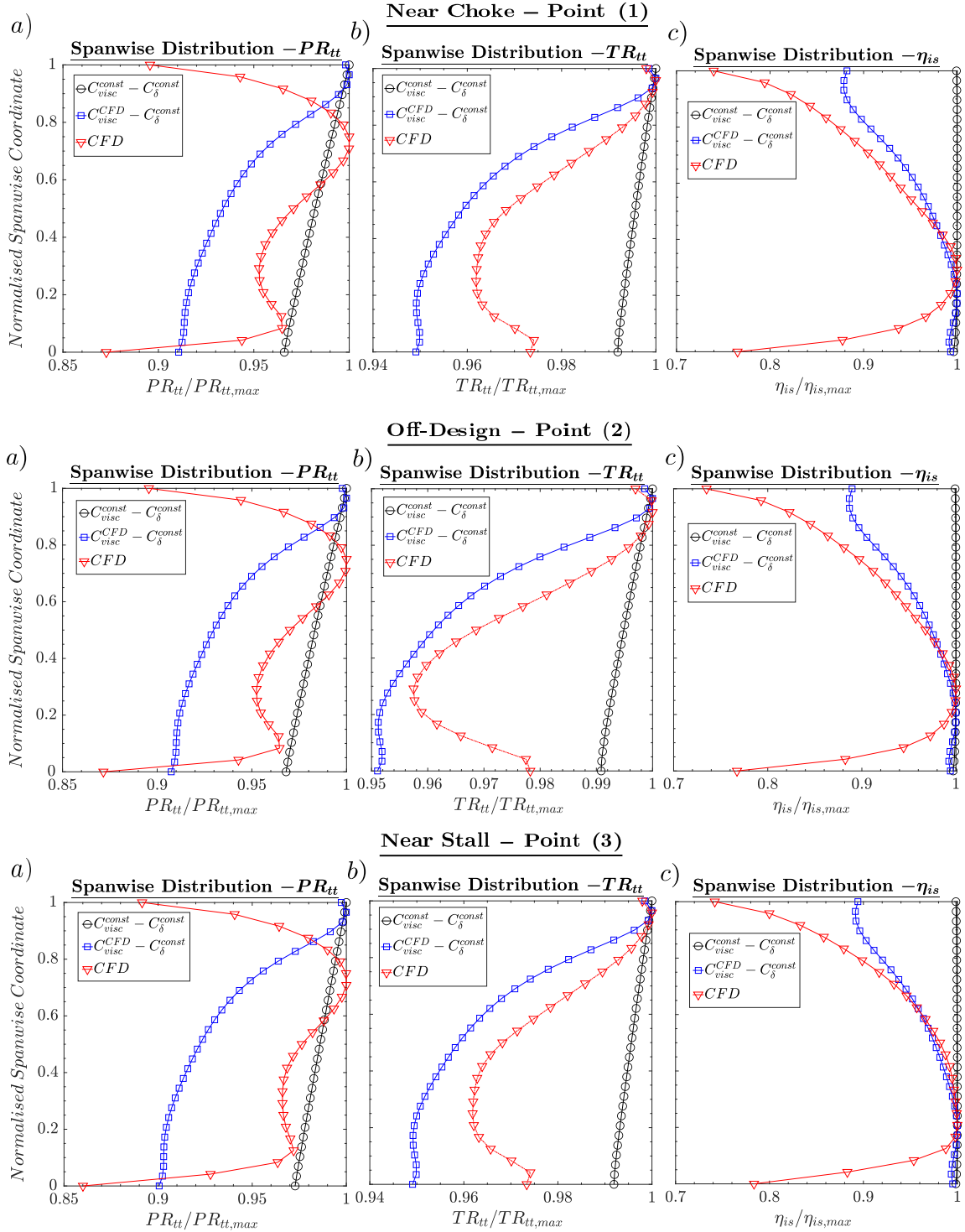


Figure 4.35: Spanwise profiles of pressure ratio, temperature ratio and efficiency of CSTAR centrifugal compressor at near choke, off-design and near stall operating conditions.

are underestimated, resulting in higher pressure ratios, lower temperature ratios and higher efficiencies. The viscous force coefficients of the impeller and vaned diffuser blades were extracted from RANS data corresponding to off-design operating points, thereby providing slightly more accurate results in the off-design region. The pressure

ratio profiles of the second model (blue) are able to more accurately capture the shape of the profile along the passage but its limitations near the shroud and the hub are evident. The sources of the pressure ratio profile errors near the shroud and the hub can be identified by examining the temperature and efficiency profiles. At the shroud, the temperature ratio profiles correlate well but the efficiency is overpredicted by the code, indicating that the source of the error is the introduction of lower losses. At the hub, the lower temperature ratio and the higher efficiency indicate that the source of the error is related to both the introduction of smaller amount of losses as well as the imposition of higher deviation angles. It is hypothesized that the introduction of a CFD-derived deviation coefficient would further improve the results, especially in the region near the hub.

4.5 Validation of blade curvature definition method

The accurate definition of the blade curvature in three dimensions is of paramount importance since it dictates, to a great extent, the imposed relative flow path and as a result, the decomposition of both the turning and the viscous force in the axial, circumferential and radial directions. The definition of the three principal angles that fully define the blade curvature are the blade, flow path and lean angle. Their definition along with the developed, vector-based estimation approach is described thoroughly in the methodology chapter (see section 3.3). The evaluation of the curvature angles is performed along distinct, camber surface meshes in three dimensions generated using piecewise cubic interpolation and an equidistant node distribution algorithm. The evaluation of the metal blockage distribution is performed by generating pressure and suction surface meshes for each blade row on the nodes of which, the local thickness of the blades are defined. To verify the validity of the curvature and blade extraction method, the curvature of rotor 37 [121] and impeller SRV2 [107] are evaluated. The distributions of the aforementioned quantities are extracted and subsequently compared against data available in the public domain [85, 107].

Figure(4.36) illustrates two different rotor 37 grids; the first is the grid used by Kottapalli [85], while the latter (right) is the meridional projection of the camber surface mesh generated by the through-flow code. A more sophisticated grid algorithm is used to generate the left grid, forming high-resolution grid regions near the LE, the TE as well as the hub and the shroud to more accurately capture the local curvature of the blade. The density of the equidistant grid generated by the code comprises 50×70 cells in the streamwise and spanwise direction respectively and

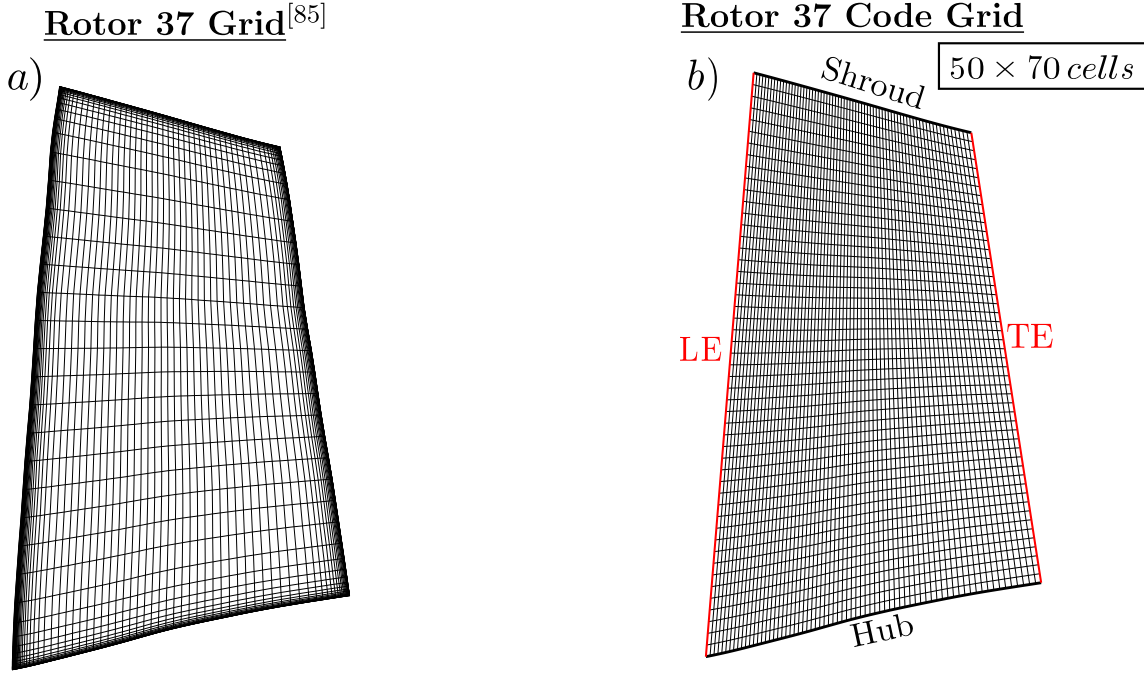


Figure 4.36: Meridional projection of Rotor 37 grid a) used by [85] and b) generated by the code.

can be adjusted to modify the streamwise and spanwise resolution of the mesh. The blade angle near the LE and TE changes progressively, thus a very high local grid density is not expected to provide any noteworthy accuracy benefits.

The curvature angles along with the metal blockage distribution of Rotor 37, estimated by [85] are depicted in figure(4.37). It is evident that the dominant angle, which dictates the turning of the flow, is the blade angle. The blade angle distribution at the LE is rather uniform with the angles progressively increasing towards the shroud, while higher variations can be observed near the TE. The flow path angle expresses the curvature of the duct in the meridional direction and is negligible near the midspan position. The effect of the flow path angles in axial compressors is more pronounced only in the vicinity of the hub and the shroud due to the convergent shape of the duct. Blade lean effects in axial compressor blades are typically small due to the presence of small lean angles in the largest part of the blade, as shown in figure (4.37c). It would be reasonable to deduce, that the influence of flow path and lean angle is expected to be of second order in axial compressors and its omission is not expected to introduce any noteworthy inaccuracies in through-flow applications. This hypothesis is nevertheless not expected to be valid to impeller blades due to the presence of higher lean and flow path angles.

Figure(4.38) shows the blade angle distributions derived by the code. The blade

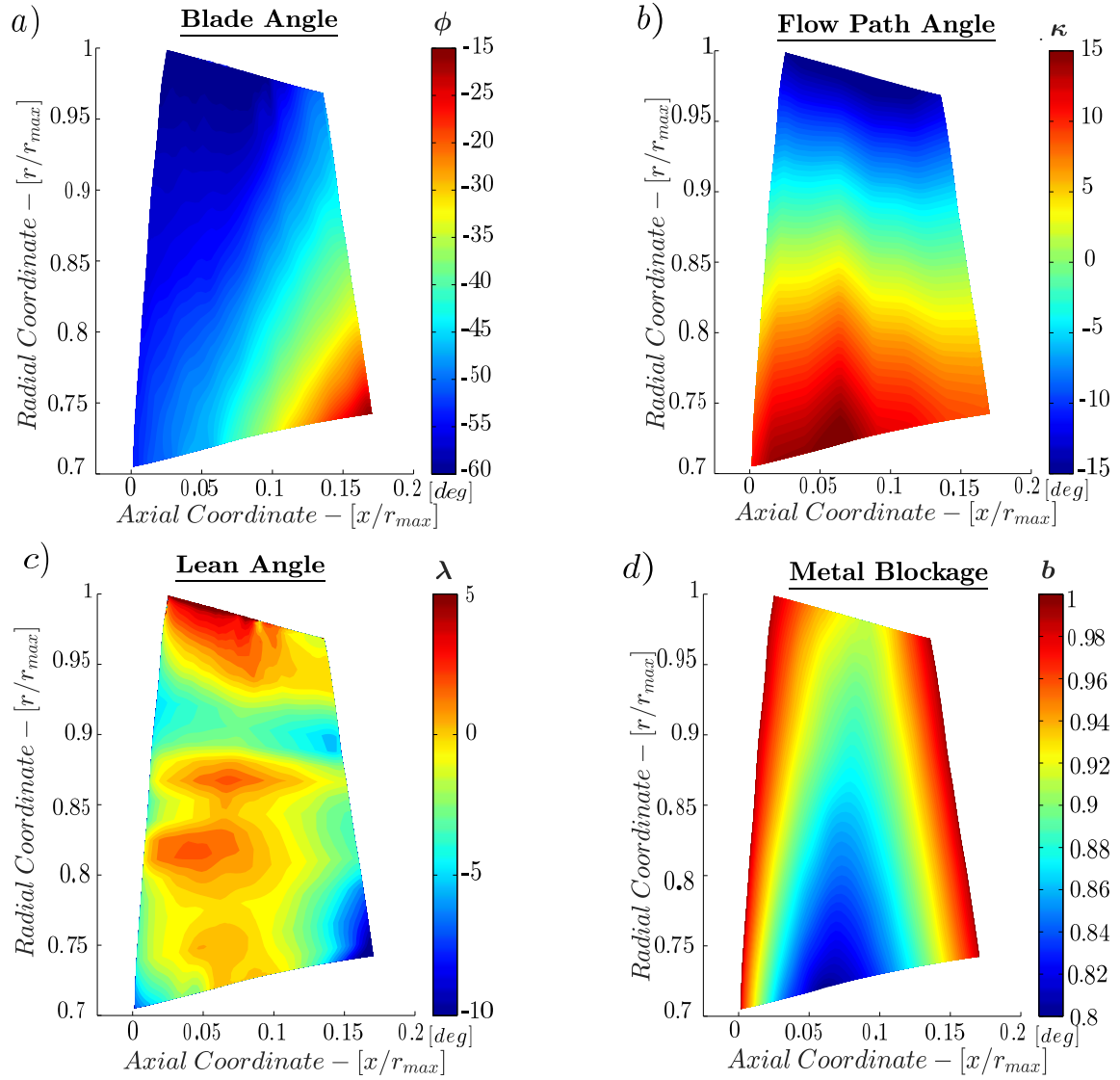


Figure 4.37: Blade angle, flow path angle, lean angle and blockage parameter distributions of Rotor 37 computed by [85].

and flow path angle distributions are in excellent agreement, while some notable differentiations are observable only in the lean angle distributions. These discrepancies are however trivial and are exacerbated due to the presence of near-zero lean angles across the majority of the blade passage. The differences in the lean angle distributions are presumably related to the raw coordinate input data as well as accumulation of interpolation errors during the geometry generation process. The blockage distributions are in good agreement, displaying similar features, however consistently higher metal blockage distributions can be seen in figure(4.37d), indicated by lower blockage parameter values. It should be noted that the lower limit of the colorbars was not set to the same value to allow for a more accurate visualization of the blockage distribution. Due to the absence of experimental data, the accuracy of

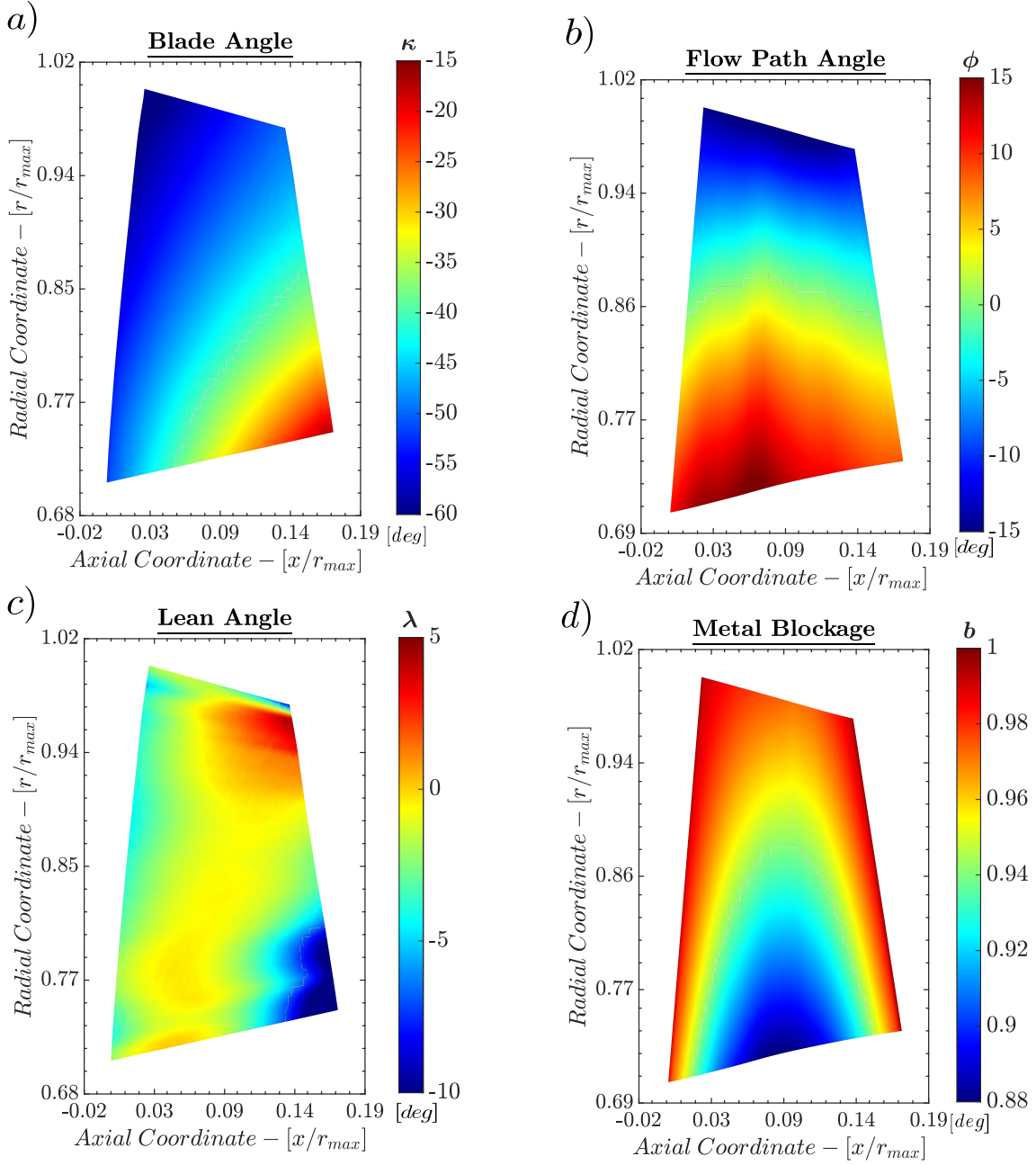


Figure 4.38: Blade angle, flow path angle, lean angle and blockage parameter distributions of Rotor 37 computed by the code.

the blockage parameter could not be further assessed, although the blockage values estimations of [85] are considered rather high for an axial compressor rotor.

The blade angle validation process is concluded by comparing the blade angle distribution of the impeller SRV2 against the blade angle data of the original experimental impeller SRV2 geometry reported in [107]. The experimental data comprise streamwise distributions of the impeller blade angles along the hub and the shroud. To assess the accuracy of the curvature definition method, a camber surface mesh

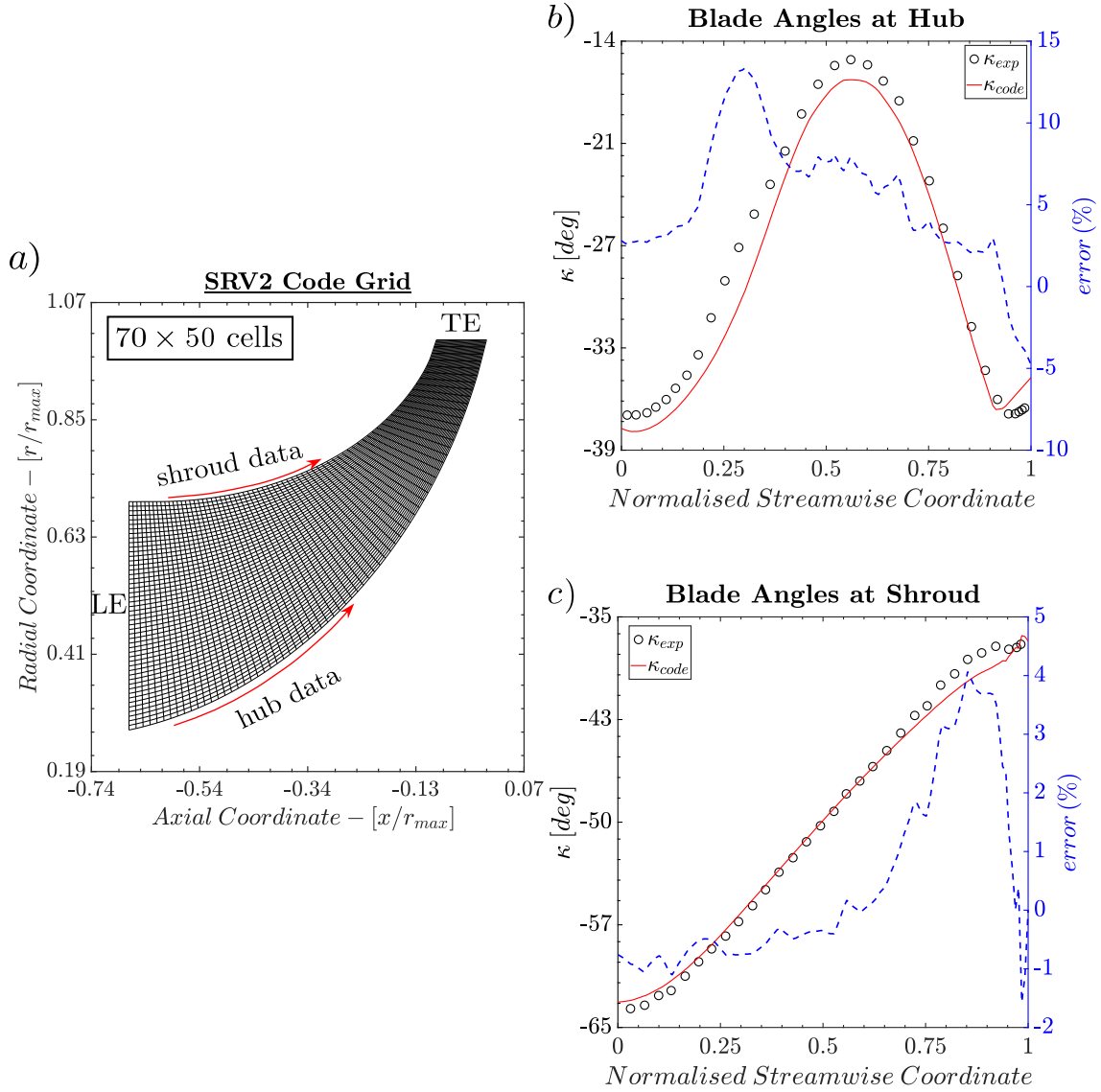


Figure 4.39: a) Meridional projection of impeller SRV2 grid generated by the code and experimental and computed streamwise distributions of blade angles at b) the hub and c) the shroud.

in three dimensions is generated. The derived distributions along the hub and the shroud are subsequently compared against the angles of the experimental impeller SRV2 geometry.

Figure(4.39a) shows the meridional projection of the camber surface mesh, consisting of 70×50 atreamwise and spanwise cells respectively. Figures(4.39b,c) show the blade angle distributions at hub and the shroud of the experimental geometry (circles) and the one computed by the code (red line) along with the corresponding errors. The original experimental blade angles are provided with respect to the tangential axis, thus they were transformed to conform with the definition used in the code, whereby

blade angles are expressed with respect to the meridional axis. The approximated blade angles at the hub correlate reasonably well with the experimental data. On average, the errors are at the order of 5%, however the overall curvature characteristics of the hub line are very accurately reproduced. Accurate blade angle estimations at TE are essential since they dictate the overall flow turning and consequently energy input into the flow. The blade angle at the hub TE is underestimated by 5%, while the error at the shroud is less than 0.01%. The results confirm the validity of the blade curvature definition method as well as the validity of the blade geometry discretisation approach, since an accurate reconstruction of the blade geometry is inextricably linked to the accuracy of the curvature extraction method.

4.6 Reverse flow modelling - Validity and limitations

The dominant flow-field features during reverse flow are primarily viscous-induced phenomena, while some of them require 3D modelling since they are inherently non-axisymmetric e.g. vortices, tip leakage flows etc. In accordance to the experimental and numerical studies presented in literature review section, aerodynamic blockage, flow deviation and losses due to mixing-out of the flow are identified as the predominant flow features, in terms of their implications on global aerodynamic performance predictions. The intensity of those flow-features is approximated transiently via the blockage-mixing model, presented in section(3.5) and introduced into the solver via the body force model and the blockage parameter. These phenomena and their implications require explicit modelling since they are viscous-related and cannot be naturally replicated in the code. The shape and consequently the accuracy of the characteristics is primarily dictated by the reverse flow modelling approach which in turn, is developed on the basis of several assumptions.

The CFD results reveal the presence of large recirculation regions upstream the impeller LE as well as highly non-uniform flow, thereby making the extraction of reliable loss and deviation data challenging (see section(5.2)). Nevertheless, CFD data extracted from 3D, steady-state, single-passage RANS simulations are used to assess the validity of certain assumptions and to the extent possible, the accuracy of the model. Aerodynamic blockage predictions from CFD simulations were not obtained, but the predictions of the new model were compared against two models in the literature using averaged flow-field quantities obtained from CFD simulations.

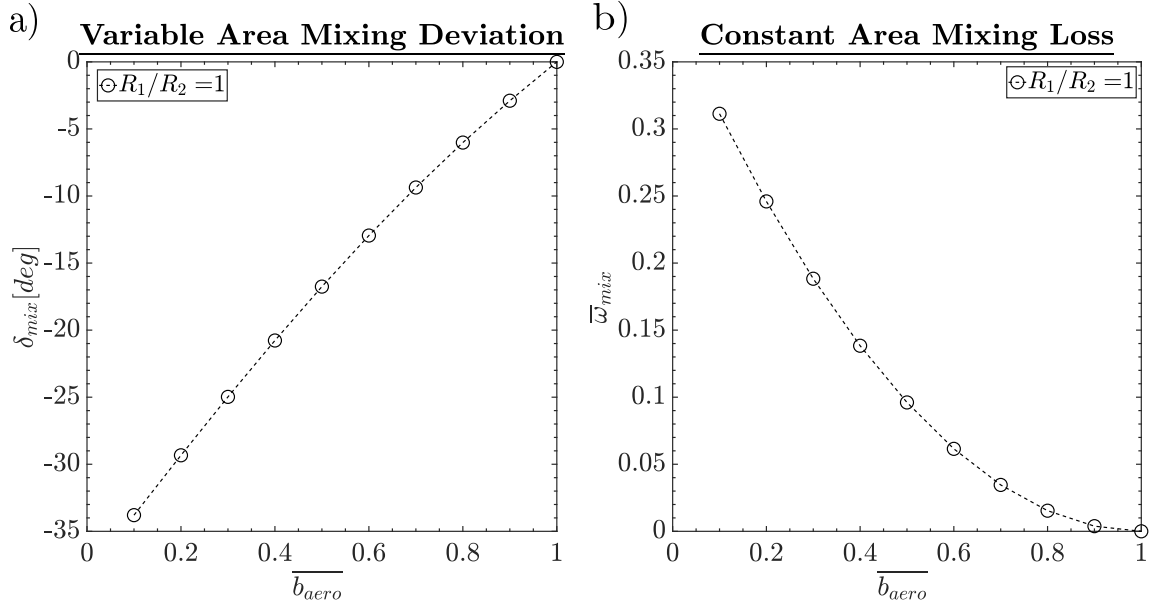


Figure 4.40: a) Deviation angle and b) loss coefficient predictions of the analytical blockage-mixing model with respect to the aerodynamic blockage due to constant area flow mixing.

Figure(4.40) shows the deviation angle and loss coefficient predictions of the blockage-mixing model due to mixing-out of the flow downstream the LE assuming constant area. The loss coefficient is negligible in the higher blockage parameter region, while even in the lower one ($\overline{b_{aero}} < 0.6$), it is of second order compared to the losses introduced by the turning force inside the blade passage.

When the flow exits the vaned diffuser passage, the area changes as a function of the radius, thus the constant area mixing assumption is invalid. Both the mixed-out flow deviation angle and the loss coefficient are a function of both the aerodynamic blockage and the change in areas designated by the radius ratio R_1/R_2 . While the selection of radius ratio is ostensibly arbitrary, it would be reasonable to set it in a way that defines the vaneless space, i.e. R_1 and R_2 corresponding to the vaned diffuser LE and impeller TE radial positions respectively. The validity of this assumption is investigated in section(4.6). To assess the sensitivity of the model with respect to the radius ratio and aerodynamic blockage, the mixed-out deviation angle and the corresponding mixing loss predictions are plotted for different blockage parameters and radius ratios (figures(4.41)).

Figures(4.41a) and (4.41b) visualise the behaviour of the deviation model with respect to the radius ratio for different blockage parameters as well as the behaviour of the model with respect to the blockage parameter for different radius ratios respectively. It can be observed that in the ~ 0.9 radius ratio range, which is

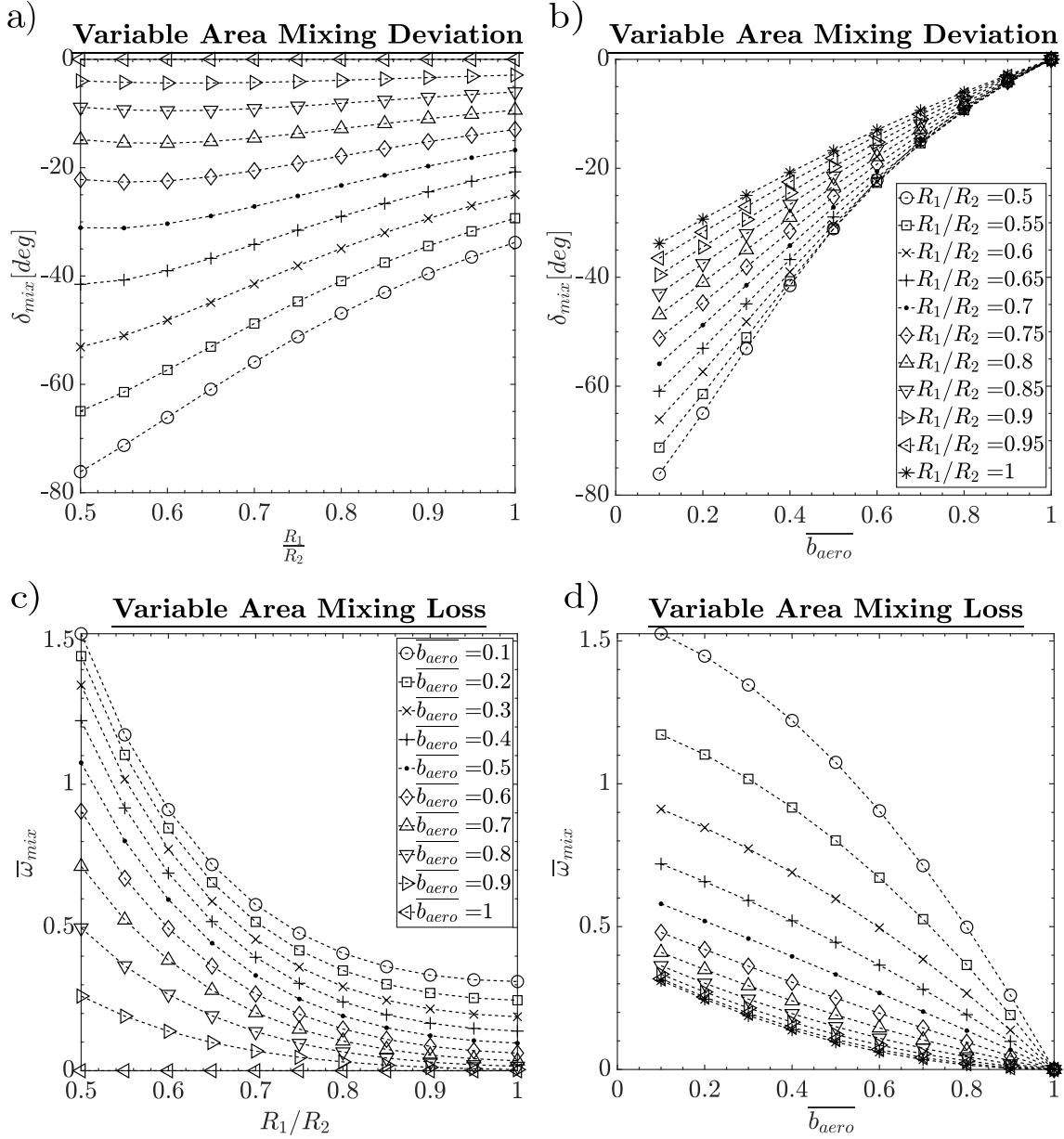


Figure 4.41: Variable area mixing a) Deviation angle against radius ratio for different blockage parameters b) deviation angle against blockage for different radius ratios, c) mixing losses against radius ratio for different blockage parameters d) mixing losses against blockage for different radius ratios.

representative of vaneless space radius ratio, flow deviation is primarily governed by the aerodynamic blockage and not the radius ratio. At lower radius ratios, the deviation angle is governed by both the aerodynamic blockage and the radius ratio defining the boundaries of the mixing region. The loss coefficient is plotted against the radius ratio and the aerodynamic blockage for different aerodynamic blockages and radius ratios respectively as shown in figures(4.41c) and (4.41d). It is evident that even though both the radius ratio and the aerodynamic blockage have a noteworthy

effect on the loss coefficient curve, the radius ratio is the dominant one. This is more prominent in the 0.9 radius ratio range, where the gradients of the loss coefficient curves with respect to radius ratio approach zero (figure(4.41c)). The variable area mixing model reduces to the constant area as the radius ratio approaches 1. For the limiting case where $R_1/R_2 = 1$, the variable area equations reduce to the constant area ones which is also evident by the identical deviation and loss coefficient curves corresponding to $R_1/R_2 = 1$.

Aerodynamic blockage

Aerodynamic blockage at different streamwise positions along the blade passage were not extracted since the post-processing of the flow-field solution was performed externally by interpolating, meridional, area/mass-averaged flow-field properties of the CFD solution on computational grids generated by the code. A rigorous extraction of the aerodynamic blockage would require the extraction and synthesis of the aerodynamic blockage at different spanwise positions in the blade-to-blade plane. This has not been performed in the present work due to time constraints but is recommended for future work. The consistency of the model is evaluated in a two-fold manner; using CFD flow-field solutions, averaged inlet and outlet properties are used to provide the required input for the model. The predictions of the model were subsequently compared against the predictions of the aerodynamic blockage models of Longley [40] and Powers et al. [63]. The latter is a function of a tuning parameter, $\hat{\alpha}$, which defines the strength of the stall and was determined experimentally in the original study. Since the stall parameter is not a pre-determined quantity, it is not known a priori, thus blockage curves are plotted for different values of the tuning parameter and compared against the predictions of Longley's and the new model. Aerodynamic blockage predictions are obtained for axial rotor, impeller and vaned diffuser geometries and their estimations are qualitatively assessed by examining the extent of the separation region along the passage.

The aerodynamic blockage predictions in the NASA rotor 37 blade passage due to flow separation at reverse flow conditions, are illustrated in figure(4.42). Blade-to-blade velocity streamlines at the lowest and highest pressure ratio of the 100% characteristic denoted with (1) and (2) respectively are also included. The blockage parameter is plotted against different incidence angles corresponding to different operating points along the characteristic, with lower incidences corresponding to higher PR settings (and higher mass flows). The Powers et al. model is plotted for a range of stall parameters, in the range of $\hat{\alpha} = [0.1 - 2]$. The stall parameter defines

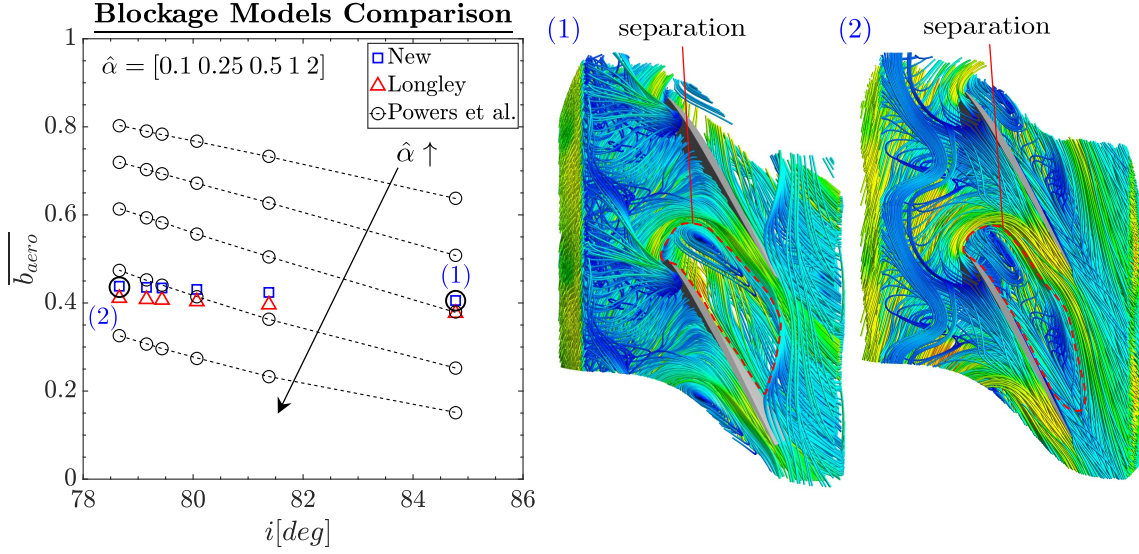


Figure 4.42: Comparison of rotor 37 aerodynamic blockage predictions at reverse flow conditions using the new, Longley’s and Powers et al. model.

the strength of the stall, thus $\hat{\alpha}$ increments result in lower blockage parameters or equivalently larger aerodynamic blockage values. In opposition to the other two models, the gradient of the Powers et al. model curves are steeper, indicating larger sensitivity with respect to the incidence angle. The predictions of the new model correlate well with Longley’s model, exhibiting a very similar behaviour and trends. The differences are presumably stemming primarily from the change in curvature at the inlet and outlet of the blade which are neglected in Longley’s model since variations in the inlet and outlet areas as well the change in rotational velocities are insignificant. The streamline contours unveil the extent of the separation bubble, the size of which, diminishes notably at lower incidence angles due to the higher momentum of the bulk flow which suppresses the separation bubble. The separation bubble is more pronounced at higher spanwise positions, towards the shroud. A qualitative assessment of the flow-field indicates that the Powers et al. model better approximates the variation of the separation bubble in this particular test case.

Aerodynamic blockage estimations were subsequently obtained for impellers SRV2 and CC3, utilizing area and mass flow-averaged data of CFD flow-field solutions as inputs for the models. The aerodynamic blockage predictions of the two impellers obtained from the 3 models along with blade-to-blade velocity streamline contours at the lowest and highest incidence angles are depicted in figures(4.43) and (4.44). The blockage estimations of the CC3 and SRV2 impellers were acquired at 40% and 60% speed respectively, with the different incidence angles effectively corresponding to a different pressure ratios, i.e. operating points on the reverse flow characteristics.

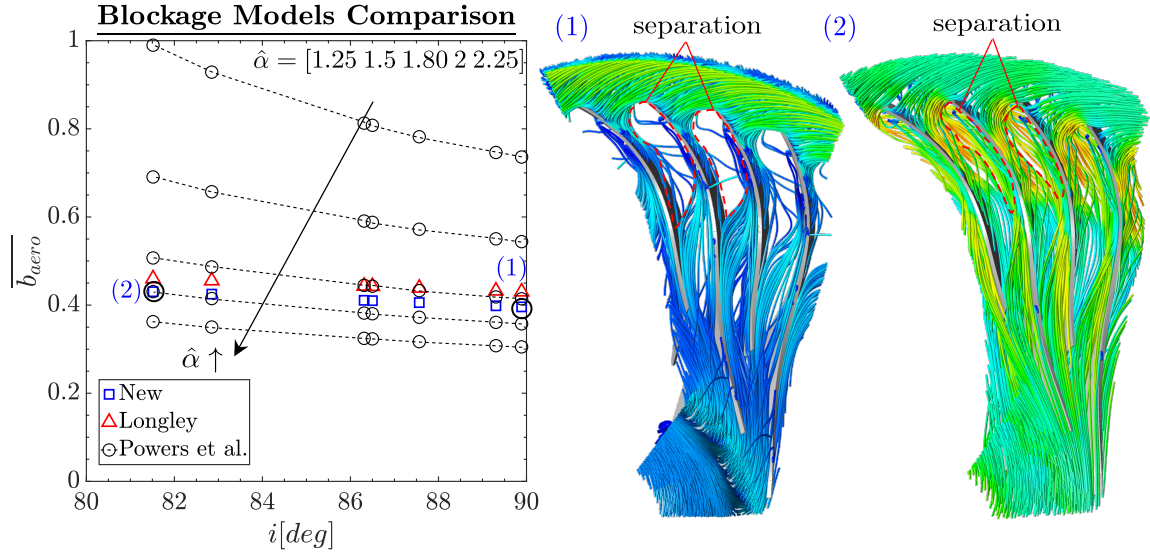


Figure 4.43: Flow-field and comparison of impeller CC3 aerodynamic blockage predictions at reverse flow conditions using the new, Longley's and Powers et al. model.

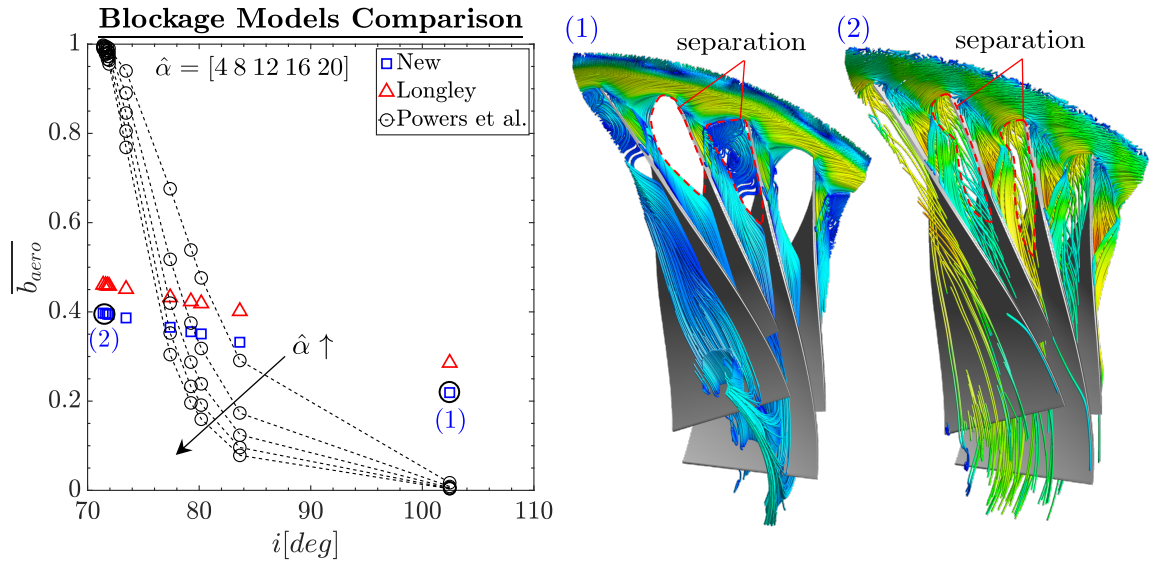


Figure 4.44: Flow-field and comparison of impeller SRV2 aerodynamic blockage predictions at reverse flow conditions using the new, Longley's and Powers et al. model.

The gradients of all blockage curves corresponding to CC3 impeller exhibit similar behaviour, predicting relatively subtle changes in aerodynamic blockage within the displayed incidence range. A large separation region forms at the TE of the impeller due to the high incidence of the incoming flow, inducing substantial aerodynamic blockage as shown by the streamline contours in figure(4.43). At higher PR settings, the larger incidence angles and the increased momentum of the incoming flow ameliorates the aerodynamic blockage effects by suppressing the separation bubble. At higher PR settings, the separation bubble manifests at higher spanwise positions

with the flow attaching significantly faster near the hub. The separation bubble is highlighted with red dashed lines and is less visible in the higher PR setting due to less flow separation near the hub. Contrary to axial compressor blades, the larger length of the impeller blades allows extensive mixing-out of the flow inside the blade passage, which is evident by the substantial reduction in the size of the separation bubble. The mixing-out of the flow along the impeller passage and the reduction of the separation bubble as the flow moves downstream the TE, is a significant modelling aspect which is taken into account via the mixing parameter which effectively reduces the predicted aerodynamic blockage. The data shown within this section are the direct output of the models and mixing effects are neglected. The flow-field in the impeller SRV2 passage exhibits identical features (figure(4.44)), however the gradient of the blockage curves derived by the Powers et al. model are remarkably steeper, predicting significant variations in the blockage parameter. The very low aerodynamic blockage predictions of the Powers et al. model in the lower incidence range as well as the excessively high blockage in the high incidence range is considered unrealistic. The new model predicts larger aerodynamic blockage compared to Longley's by approximately 6.7% along the entire characteristic.

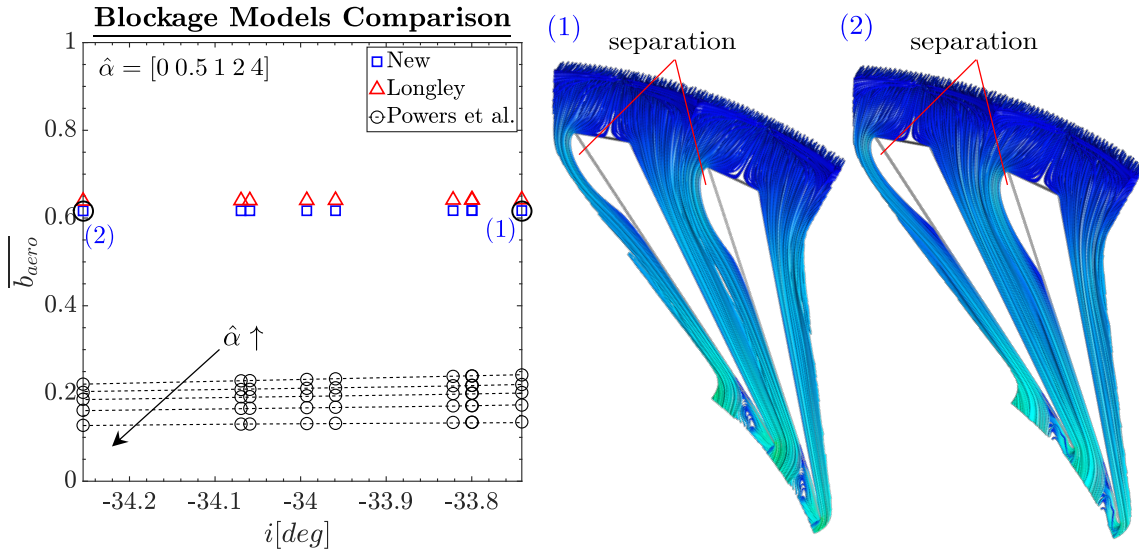


Figure 4.45: Flow-field and comparison of NASA CC3 vaned diffuser aerodynamic blockage predictions at reverse flow conditions using the new, Longley's and Powers et al. model.

The applicability of the models to vaned diffuser blades was subsequently investigated utilizing CFD flow-field solution data of NASA CC3 compressor with vaned diffuser. The aerodynamic blockage predictions are plotted against the incidence angle at the TE, along with absolute velocity streamlines corresponding to the highest and lowest incidence angles (figure(4.45)). Despite the large changes in pressure

ratio (2.26 - 3.64), the extent of the separation bubble exhibits negligible changes in its shape and size, indicating that the incidence angle is the quantity that is primarily dictating the flow features of the separation bubble. The predictions of Longley's model exhibit similar trends to the new model, which estimates slightly higher aerodynamic blockage compared to Longley's model (lower b parameter). The predictions of the model indicate, that the implications of the large changes in the inlet and outlet areas on the aerodynamic blockage predictions are of second order compared to the incidence angle of the incoming flow. The Powers et al. model predicts unrealistically high aerodynamic blockage even when the stall parameter, $\hat{\alpha}$ is set to zero. This is evident by the absolute velocity streamline contours which clearly visualise the shape and size of the separation bubble.

Mixing - Deviation

Once the flow exits the blade, the mixing-out of the jet stream with the separated flow changes the circumferential momentum, resulting in a larger mixed-out flow angle. The change in flow direction due to mixing mechanisms is estimated by a new, simplified, analytical model which in turn is derived on the basis of a zero-deviation angle at the LE. This assumption was assessed in different compressor blade geometries at reverse flow conditions to evaluate its validity. The flow-field solution was obtained from 3D, single passage, steady-state CFD results at reverse flow conditions. The flow-field was subsequently circumferentially area or mass flow-averaged and interpolated onto the centroids of the meridional grid generated by the through-flow code. Since the blade angle distribution along the through-flow grid is known, the local deviation angle is derived by subtracting the flow angles from the blade metal angles.

Figures(4.46) and (4.47) illustrate the deviation distribution of the impeller CC3 (vaneless diffuser configuration) and impeller SRV2 at different operating points at 40% and 60% speed respectively. In both cases, the flow enters the blade at the TE with an excessive incidence angle which precipitates flow separation primarily in the vicinity of the TE. The flow near the hub reattaches rapidly compared to the shroud, which is evident by the deviation contours shown in figures(4.46) and (4.47). The spanwise non-uniformity of the blockage near the TE at higher pressure ratio settings was discussed in the previous section and captured by the streamline contours in figures(4.43) and (4.44). The area-averaged relative flow angles along the impeller passage indicate that the flow reattaches approximately 20% downstream the TE in the streamwise direction with the flow remaining attached to the blade

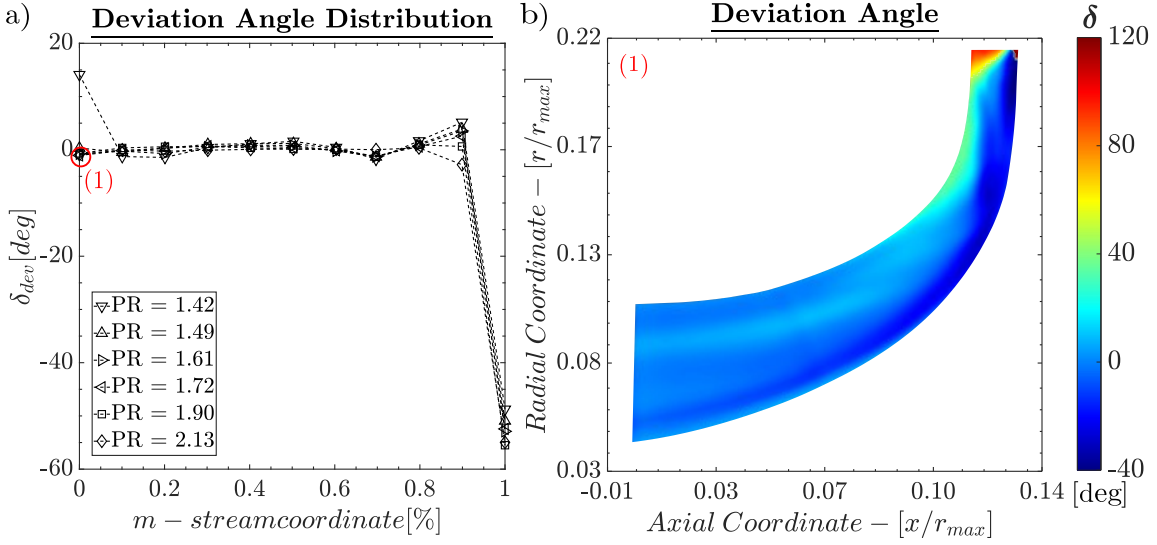


Figure 4.46: a) Area-averaged deviation angle at different streamwise positions in NASA CC3 impeller and b) deviation angle contour at point (1), corresponding to a high pressure ratio setting.

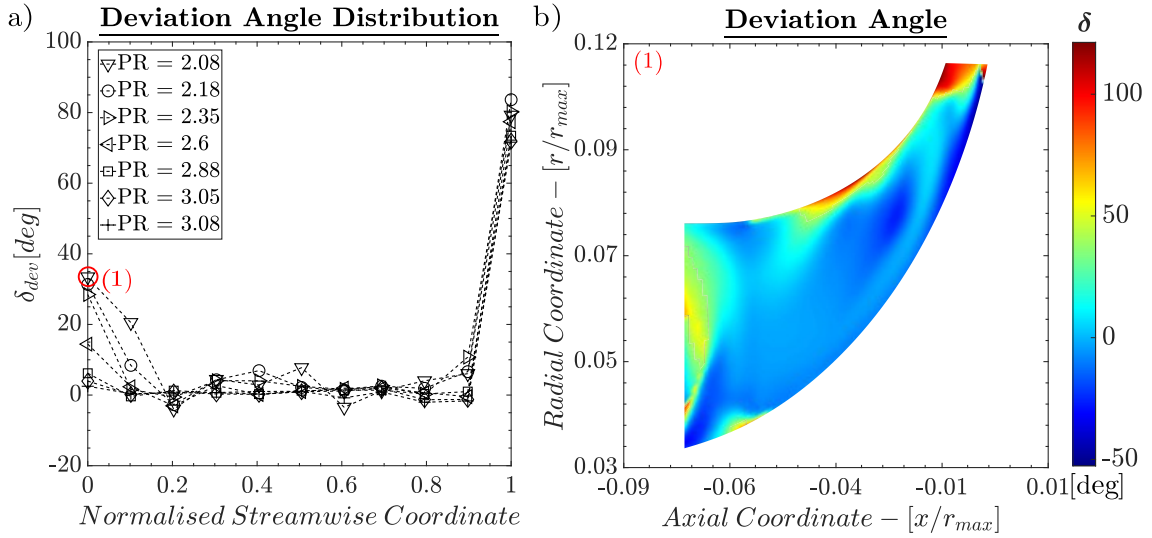


Figure 4.47: a) Area-averaged deviation angle at different streamwise positions in SRV2 impeller and b) deviation angle contour at point (1), corresponding to a low pressure ratio setting.

until it exits the TE. Throughout the entire characteristic, the flow exits the passage at the TE at a nearly zero deviation angle, while a high deviation angle at the LE is only recorded at the lowest PR setting. This occurs due to a recirculation region manifesting at the inlet of the compressor which impedes the flow from exiting the passage; at lower PR settings, the recirculation region becomes more pronounced, while the lower momentum of the incoming flow in the relative FoR makes it more susceptible to flow misalignments (see section(5.2)). This is exacerbated due to the

absence of the splitter blades near the LE which results in a sudden increase in the blade pitch and consequently reduced flow guidance.

The impeller SRV2 incorporates an inlet bell-mouth which amplifies the recirculation region, thereby exacerbating flow misalignment. At higher PR settings, the recirculation region is more suppressed and the higher-momentum flow in the impeller exits the blade at the LE with less than 5° deviation. As the PR reduces, the flow misalignment at the LE increases, resulting in high deviation angles up to approximately 33° (figure(4.46)). In the low PR region, flow misalignment initiates near the splitter blade LE ($m \approx 0.19$) due to the sudden change in the impeller pitch, with the flow misalignment being more pronounced in the lower spanwise positions where blockage effects are more intense. The results indicate that at higher PR settings, the deviation is negligible, while even at lower ones, the deviation angles primarily manifest due to the presence of a recirculation region downstream the LE, in the inlet region. This hypothesis is verified by the consistently low deviation angles at the NASA CC3 impeller LE throughout the entire operating range.

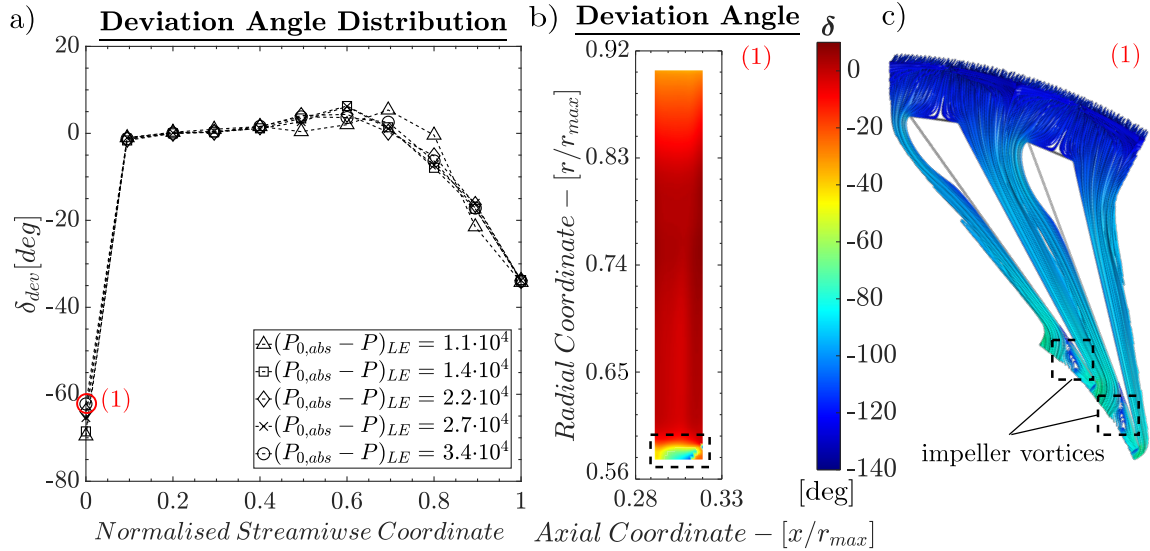


Figure 4.48: a) Area-averaged deviation angle at different streamwise positions in NASA CC3 vaned diffuser, b) deviation angle contour and c) absolute velocity streamlines at point (1), corresponding to a high inlet dynamic head.

A similar analysis is performed to investigate the zero-deviation assumption in vaned diffuser blade passages. The curves shown in figure(4.48) correspond to different inlet dynamic heads. Area-averaged relative flow angles are obtained across the blade passage and are used to extract the average deviation angles in the vaned diffuser passage of the NASA CC3 compressor. The results indicate the presence of a large misalignment region in the vicinity of the TE, due to flow separation. The flow

fully attaches to the blade approximately 30% downstream the TE in the streamwise direction and remains perfectly aligned with the blade until the vicinity of the TE, where the flow deviates from the blade camberline substantially. The flow detachment at that location is caused by an impeller-induced vortex which is convected upstream the vaneless space, generating blockage at the inlet of the vaned diffuser, which in turn causes flow misalignment (see figure(4.48)).

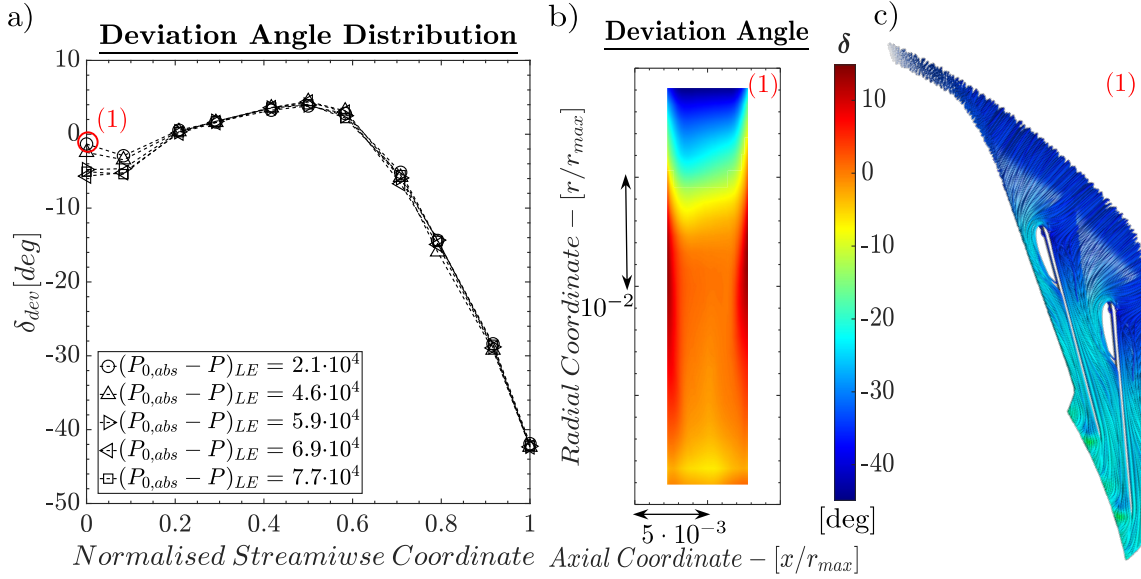


Figure 4.49: a) Area-averaged deviation angle at different streamwise positions in CSTAR vaned diffuser, b) deviation angle contour and c) absolute velocity streamlines at point (1), corresponding to a high inlet dynamic head.

To verify the hypothesis that in the absence of vortex structures, the deviation of the flow would effectively be negligible, the flow misalignment of the CSTAR vaned diffuser passage was extracted for different inlet dynamic heads. The impeller is also modelled, therefore a degree of interaction between the impeller and the vaned diffuser is expected. In figure(4.49c), a separation bubble can be observed near the TE. At approximately 40% downstream the TE in the streamwise direction, the flow fully reattaches to the blade and exits the blade with a deviation angle at the TE that does not exceed 6° even in the lowest inlet dynamic head range. During reverse flow operation, the diffuser operates as a convergent nozzle, accelerating the flow, thus no high adverse pressure gradients are present in the duct, preventing flow detachment. The results seem to verify the hypothesis that at reverse flow conditions, the deviation angle is expected to be negligible and that the deviation is expected to be primarily the outcome of other flow-field phenomena, located downstream the TE of the blades.

The validity of the negligible deviation angle at the LE during reverse flow

conditions is finally evaluated in an axial compressor rotor geometry (rotor 37). The curves correspond to different operating points along the design speed reverse flow characteristic and indicate that the deviation angle is negligible regardless of the imposed PR boundary conditions. The large flow misalignment near the TE diminishes progressively and the flow aligns with the blade curvature approximately 40% downstream the TE. Except for a small region at the LE, near the shroud, the flow exits the passage with minor deviation angles at the LE.

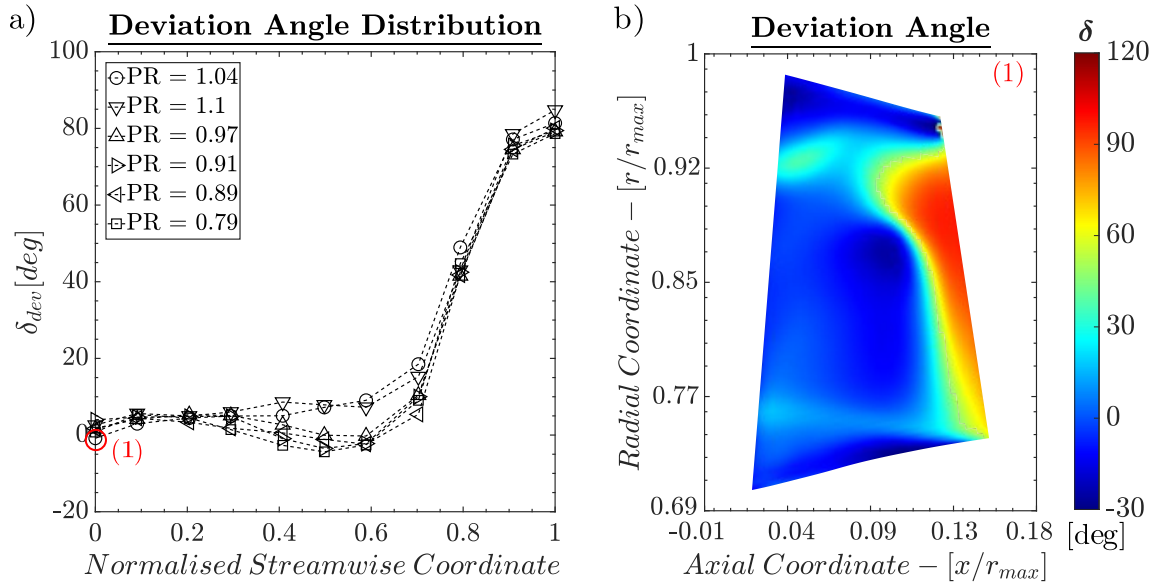


Figure 4.50: a) Area-averaged deviation angle at different streamwise positions in rotor 37 passage and b) deviation angle contour at point (1), corresponding to a high pressure ratio setting.

The results verify the assumption on the basis on which, the model was derived, namely that during flow reversal, deviation is expected to be trivial. During flow reversal, compressor blades act like nozzles, accelerating the flow in the relative FoR rather than decelerating it. Due to the favourable pressure gradient, the flow rapidly aligns with the local camberline despite the large incidence angles and the flow separation at the TE, exiting the blade with minor deviation angles. Impeller blades are more prone to flow deviation due to the absence of additional guidance near the LE that splitter blades provide, nevertheless the results designate that the origin of flow deviation at the TE is a phenomenon induced by complex phenomena downstream the TE. Due to the presence of large recirculation regions at the inlet and the highly non-uniform flow-field in the vaneless space, reliable deviation data downstream TE of the blades could not be extracted, thus the accuracy of the mixed flow angles downstream the TE of the blades could not be verified.

Mixing - Losses

During reverse flow conditions, the loss model introduces losses in a two-fold manner; via the turning force, by projecting the turning force normal to the blade geometry and via a viscous force, exerted in ducted domains to account for incomplete flow mixing once the flow exits the blade passage (only at reverse flow conditions). The first causes the flow to separate at the TE and introduces losses in the vicinity of the TE, since the flow is locally misaligned and the turning force is not exerted perpendicular to the relative flow-path but rather perpendicular to the local camberline. The latter introduces additional mixing losses, predicted via the simplified, analytical constant-area or variable-area mixing model. These generally are of second order compared to the losses introduced by the turning force. A skin friction loss is also imposed in the blade passage using Aungier's skin friction correlation [68] but its implications are also of second order.

To assess the validity of the loss model, losses in blade passages are extracted and quantified by means of rotary stagnation pressure losses. The use of rotary stagnation pressure is preferred over the relative FoR one, due to its independence on the rotational velocity caused by changes in the blade passage curvature in the meridional plane, expressed by the flow path angle, ϕ . It is a quantity which in an isentropic flow remains invariant, therefore rotary stagnation pressure decrements manifest purely at the presence of dissipative flow phenomena. The losses are extracted from steady-state, CFD, reverse flow simulations and are subsequently compared against the ones extracted from through-flow simulations (imposed as described above) to assess the validity and accuracy of the loss model.

Figure(4.51) shows the streamwise distribution of the average rotary stagnation pressure loss in the impeller SRV2 passage at reverse flow conditions. The curves correspond to different operating points on the 60% reverse flow characteristic. The flow enters at the TE and exits from the LE, denoted by a streamwise coordinate 1 and 0 respectively. The CFD data indicate that the losses are predominantly occurring near the TE, where the separation bubble forms (figure(4.51b)). The high-loss region extends approximately up to 20% downstream the TE, which qualitatively is in agreement with the separation bubble size illustrated in figure(4.44). Downstream the separation region, losses remain effectively constant with notable rise in pressure losses recorded in the lower PR range as the flow approaches the LE. This occurs due to flow separation near the LE caused by the enlargement of the recirculation region, which becomes more pronounced as the imposed PR diminishes. The losses introduced

via the turning force model are capable of recreating the local flow separation and the corresponding losses with reasonable numerical and physical accuracy. The high-loss region is shorter by approximately 10% and the imposed losses in through-flow simulations are on average higher than the CFD ones.

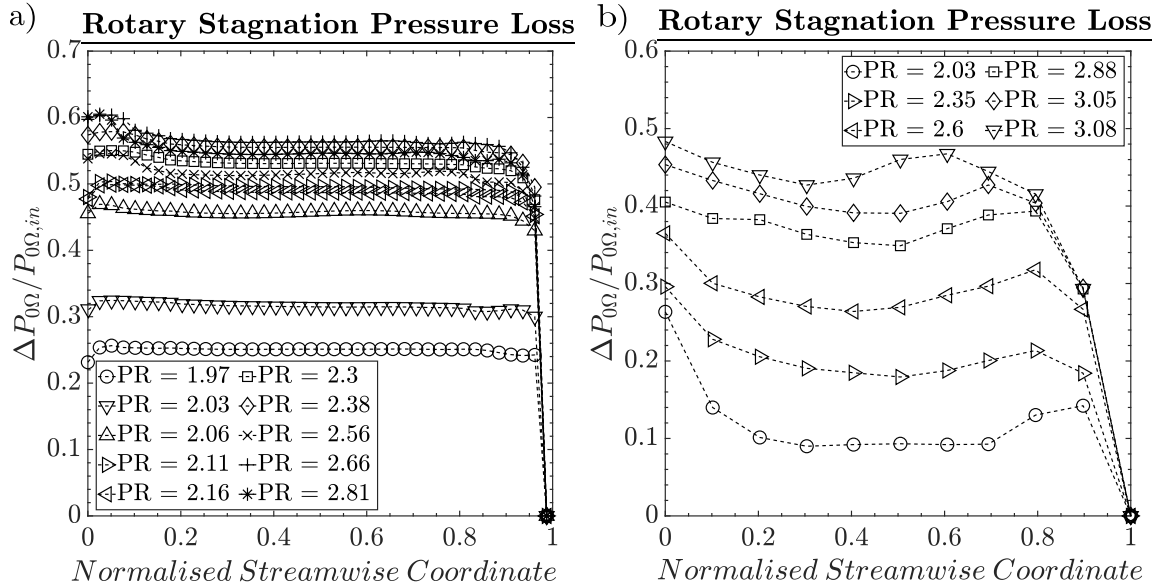


Figure 4.51: Comparison of streamwise rotary stagnation pressure loss distribution in impeller SRV2 passage at reverse flow conditions and 60% speed obtained a) from the code and b) from CFD simulations.

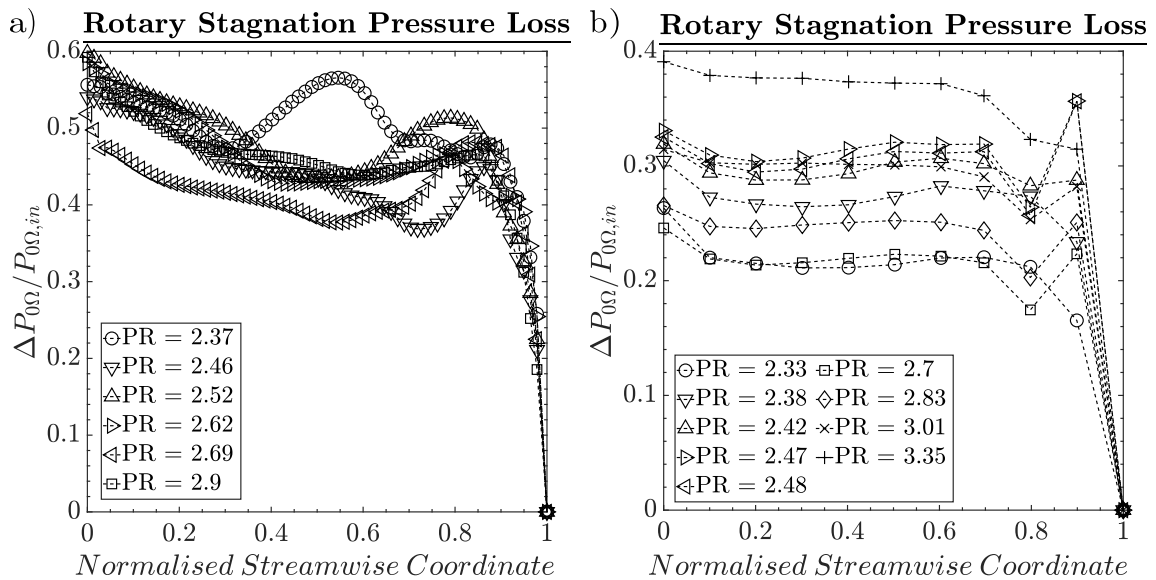


Figure 4.52: Comparison of streamwise rotary stagnation pressure loss distribution in impeller CC3 passage at reverse flow conditions and 80% speed obtained a) from the code and b) from CFD simulations.

The loss distribution along the NASA CC3 impeller blade passage, obtained by the through-flow code and the steady-state RANS simulations at 80% design speed are shown in figure(4.52). The loss distribution exhibits an analogous behaviour to that of impeller SRV2; a high-loss region coinciding with the presence of the separation bubble, extending up to approximately 30% downstream the TE is observed in figure(4.52b). Downstream the separation region, the losses do not display any noteworthy variations apart from the region near the LE, where a relatively minor increase in the pressure loss is shown. The loss model captures the aforementioned trends exceptionally well but the losses are consistently overestimated along the entire reverse flow characteristic.

To assess the applicability of the model to vaned diffuser blades, the NASA CC3 vaned diffuser passage is used to perform a similar comparison. Since the blade passage is stationary, the losses are reported in terms of absolute stagnation pressure losses, while the operating points in terms of inlet dynamic heads. Streamwise pressure loss distributions obtained by the through-flow code and CFD simulations are depicted in figures(4.53) and (4.54). The graphs correspond to different operating points on the 60% and 80% characteristics respectively.

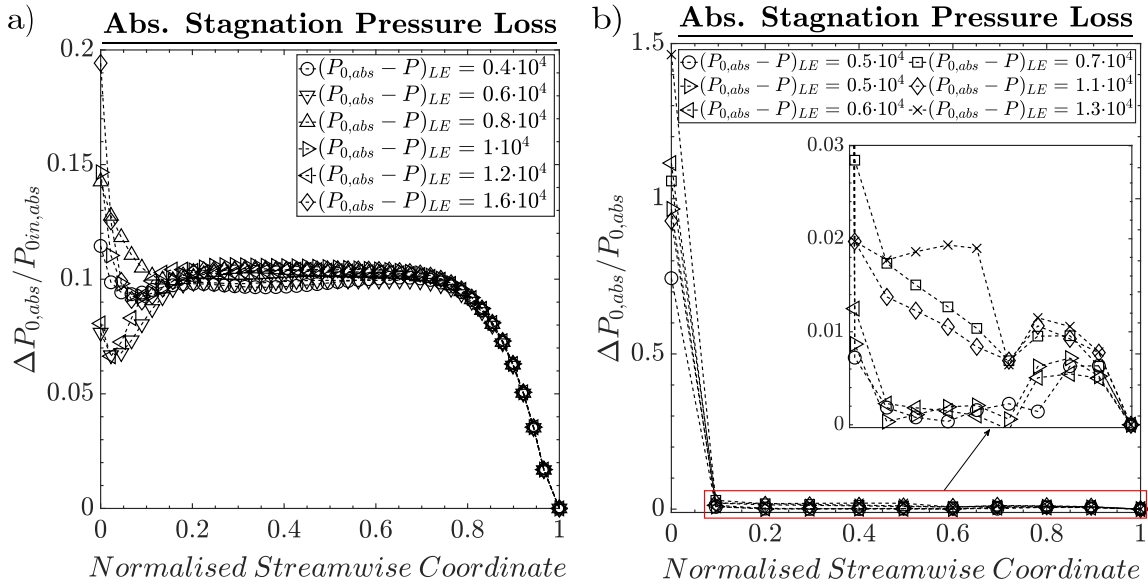


Figure 4.53: Comparison of rotary stagnation pressure loss distribution in the streamwise direction in vaned diffuser CC3 at reverse flow conditions and 60% impeller speed obtained a) from the code and b) from CFD simulations.

The pressure loss predictions of the CFD model show a sharp rise in pressure losses near the TE followed by a constant (lower dynamic heads) or rather linear increase (higher dynamic heads) in losses downstream the initial, high-loss region. The initial, high-loss region is the outcome of the flow-separation near the TE, while

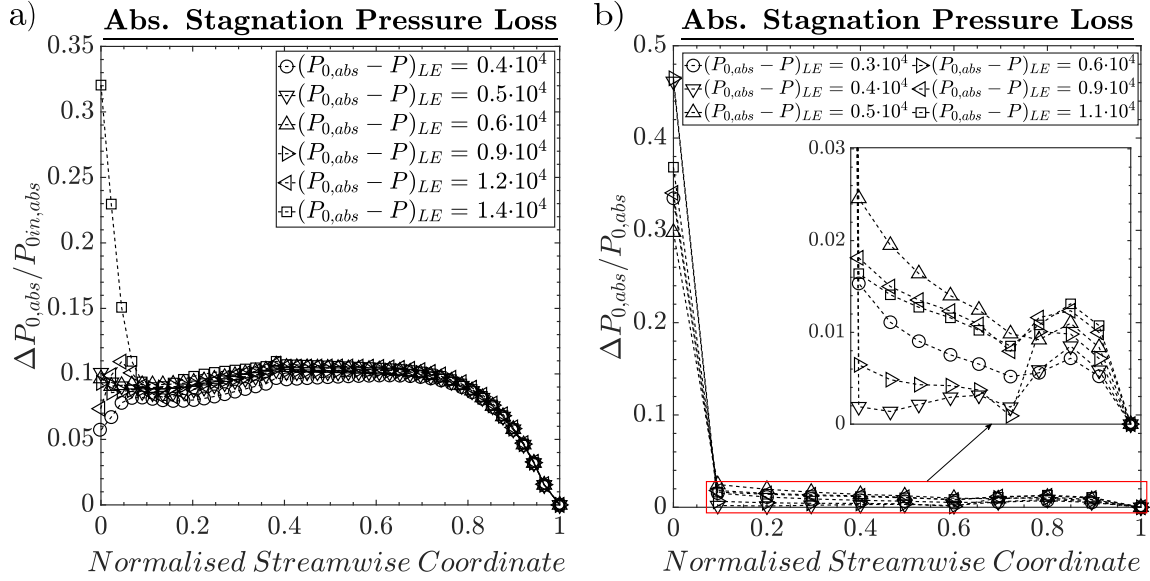


Figure 4.54: Comparison of rotary stagnation pressure loss distribution in the streamwise direction in vaned diffuser CC3 at reverse flow conditions and 80% impeller speed obtained a) from the code and b) from CFD simulations.

the subsequent increase in the losses could be originating to enhanced mixing losses and wall shear stresses. Mixing in axial stator blades is occurring at a faster rate compared to rotor blades, as experimentally reported by Gamache and Greitzer [59]. Since extensive mixing phenomena take place in long blades such as impellers and vaned diffusers, the increase in losses downstream the separation region is presumably related primarily to mixing. Excessive losses are predicted by the CFD model near the LE; this is caused by flow separation due to upstream convection of vortex structures from the impeller. This impeller-diffuser interaction is replicated reasonably well by the code which is evident by the increase in losses near the LE in figures (4.53a) and (4.54a). The losses introduced via the turning force model along the blade passage are substantially higher but the excessive losses due to the highly separated flow at the inlet are underpredicted. The implications of the misestimations of losses in vaned diffuser passages are expected to be of second order and potentially trivial in terms of global aerodynamic performance predictions since the change in the conditions at the physical inlet are predominantly dictated by the losses of the rotor and the forces it exerts, opposing the incoming reverse flow.

Apart from the losses introduced via the turning force, additional losses are introduced to account for incomplete flow mixing once the flow exits the blade passage. Due to the presence of recirculation regions in the inlet duct region, downstream the impeller (reverse flow) and the non-uniform flow-field in the vaneless space, loss measurements were not deemed satisfactorily reliable. Nevertheless, a comparison

between the predictions of the model and the CFD simulations is carried out to perform a preliminary investigation of the model's validity.

Figure(4.55) shows a comparison of the mixing loss coefficient between CFD and the analytical mixing model for the NASA CC3 impeller (left) and the vaned diffuser (right) at 80% design speed. The loss coefficient is obtained by dividing the absolute pressure loss in the duct upstream the impeller as well as in the vaneless space by the dynamic head just upstream the impeller and the vaned diffuser blade TE respectively. The predictions of the model are obtained using area and mass-flow averaged properties at the inlet and outlet of the ducted domains from the corresponding CFD simulations. The radius ratio of the variable-area mixing model used to predict the loss coefficient in the vaneless space is set to the radius ratio defined by the geometrical boundaries of the vaneless space. Since the mixing loss model predictions are primarily a function of the incidence angle, the data are plotted against the incidence angle. The estimations of the mixing loss coefficient at the inlet duct (figure(4.55a)) indicate that the model is capable of providing reasonable estimations of the mixing losses, especially in the higher incidence / lower PR region. At higher PR settings, the predictions seem to deviate more from CFD predictions which could potentially be associated with compressibility effects (the model assumes incompressible flow), however the uncertainty in the data extraction process does not allow for conclusive interpretation of the results.

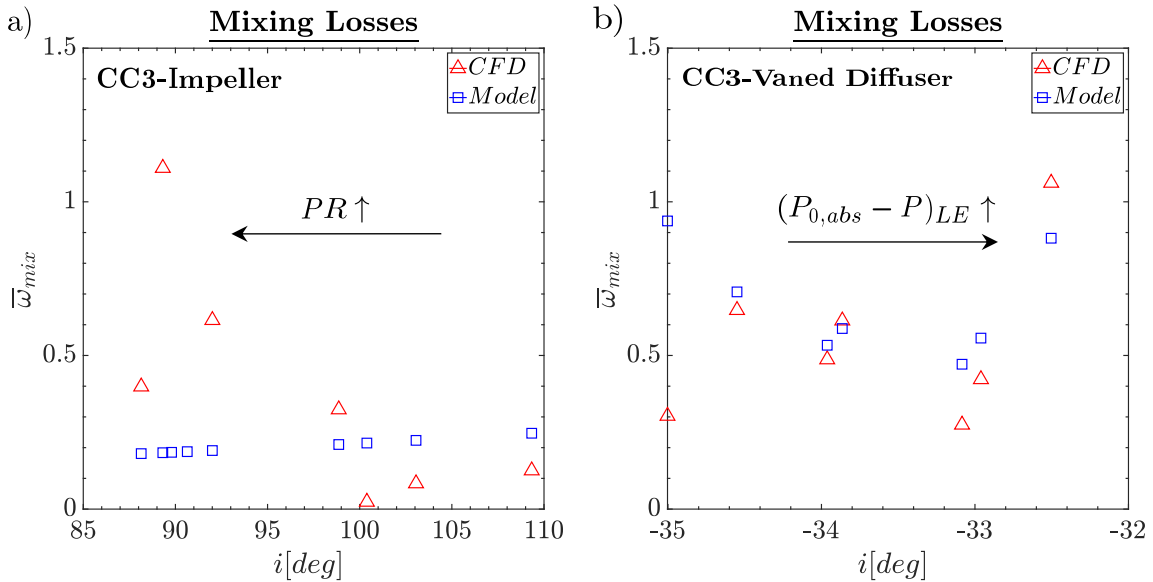


Figure 4.55: Mixing losses obtained by CFD simulations compared by predictions of the analytical model a) at the inlet duct and b) at the vaneless space of NASA CC3 compressor at reverse flow conditions.

The loss predictions in the vaneless space of the CC3 compressor are shown in figure(4.55b). The predictions of the model are in very good agreement with the measured pressure loss coefficient throughout the entire characteristic. The incidence angle variations with respect to the dynamic head at the TE are rather minuscule since no upstream bladed components are present. A large error can only be seen in the lowest PR setting (highest negative incidence) but is presumably related to the high non-uniformity of the flow in the vaneless space which is exacerbated at lower PR settings. The results indicate that the predictions of the analytical mixing model are providing reasonable estimations of the losses downstream the LE, however further investigation is required.

4.7 Blockage-mixing modelling implications

The precision of the steady-state global aerodynamic performance predictions at reverse flow conditions is quantified hereafter primarily by means of pressure ratio (total-to-total and total-to-static) and torque (actual and/or torque parameter) characteristics. In this section, the implications of the aforementioned flow features on global aerodynamic performance predictions are assessed individually, while CFD results are used to assess the validity of the models. The evaluation of those implications also provides a better understanding regarding the identification of the sources causing discrepancies between CFD and the through-flow predictions in the next chapter.

Aerodynamic blockage effect

At reverse flow conditions, the excessive incidence of the incoming flow at the TE of the blade results in the formation of a large separated region, which depending on the length of the passage and the intensity of the mixing, occupies a fraction and potentially the entire passage in the streamwise direction. The reduction in the effective area due to aerodynamic blockage effects is expected to substantially influence the choking mass flow at reverse flow conditions, but not the energy transfer along the characteristic.

To assess the implications of aerodynamic blockage on global aerodynamic performance, the 60% design speed reverse flow characteristic of the SRV2-0 centrifugal compressor is obtained by imposing 0%, 20% and 40% aerodynamic blockage corresponding to $\overline{b_{aero}} = 1$, $\overline{b_{aero}} = 0.8$ and $\overline{b_{aero}} = 0.6$ respectively. The modified curvature region is set to 10% of the passage's meridional length and losses are only

introduced via the turning force; additional mixing losses downstream the duct as well as flow deviation due to flow mixing are considered negligible. For reference, the results are compared against the reverse flow characteristic obtained by steady-state, RANS simulations using the SRV2-0 model presented in section(5.1.3) and open boundary conditions at both the inlet and the outlet.

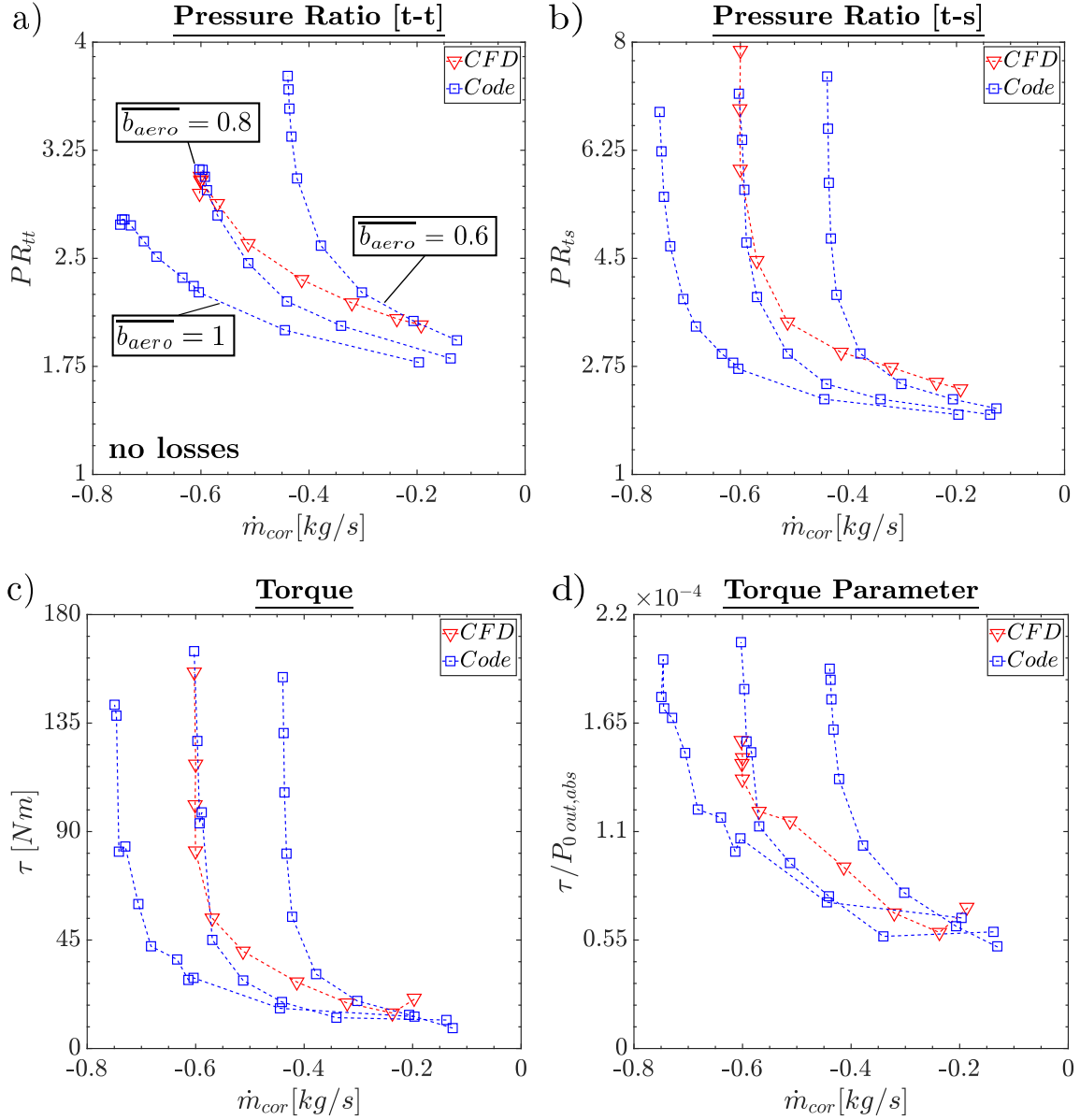


Figure 4.56: Comparison of a) total temperature ratio, b) total pressure ratio and c) flow angle spanwise distribution at the outlet obtained by CFD simulations and the code using CFD and uniform coefficients.

Figure(4.56) shows the total-to-total and total-to-static pressure ratio characteristics along with the torque and torque parameter characteristics, with the latter being the torque normalised by the absolute total pressure at the physical outlet (flow inlet

at reverse flow). It is evident that even though aerodynamic blockage drastically alters the choking mass flow capacity of the compressor and therefore the gradient of the characteristic, it is not directly affecting the displayed quantities. Aerodynamic blockage does not interfere in the energy exchange process unless mixing effects and primarily changes in the circumferential momentum are considered in the analysis (neglected here). This is more pronounced in the lower mass flow region, where all quantities tend to collapse into a single characteristic. It should be noted that the 20% blockage characteristic ($\overline{b_{aero}} = 0.8$) is in very good agreement with the CFD characteristic, especially in the higher mass flow region achieving a noteworthy improvement compared to the baseline, zero-blockage characteristic ($\overline{b_{aero}} = 1$).

Flow deviation effect

Following a similar approach, the effect of flow deviation on the aforementioned quantities due to mixing effects is investigated by applying a prescribed amount of aerodynamic blockage and comparing the characteristics with and without flow deviation. The mixed-out relative flow angle and the corresponding deviation is estimated by the constant-area, blockage-mixing model. Losses are introduced only via the turning force and the modified curvature region is set to 10%. Two baseline characteristics are initially obtained, by imposing a 20% and 40% aerodynamic blockage, losses via the turning force but without deviation and additional mixing losses. Characteristics are subsequently obtained by imposing the mixed-out deviation angle and compared against the baseline ones.

The pressure and torque characteristics corresponding to the four different numerical configurations are shown in figure(4.57). Mixing-out of the flow causes the flow to decelerate as the bulk flow mixes with the separated flow, thereby increasing the relative flow angle as the flow moves towards the LE of the blade. This essentially increases flow turning, thereby resulting in augmented energy input into the flow and in turn, higher pressure ratio and reduced mass flow capacity. Since the direction of the mixed-out flow is a function of the aerodynamic blockage, larger flow deviation is predicted by the model, which is in turn is reflected on the larger changes in the $\overline{b_{aero}} = 0.8$ characteristic when deviation is imposed. It is evident that due to the two-fold effect of the deviation angle, i.e. reduced mass flow capacity and additional energy input, changes in flow direction due to mixing constitute one of the most essential aspects of reverse flow modelling.

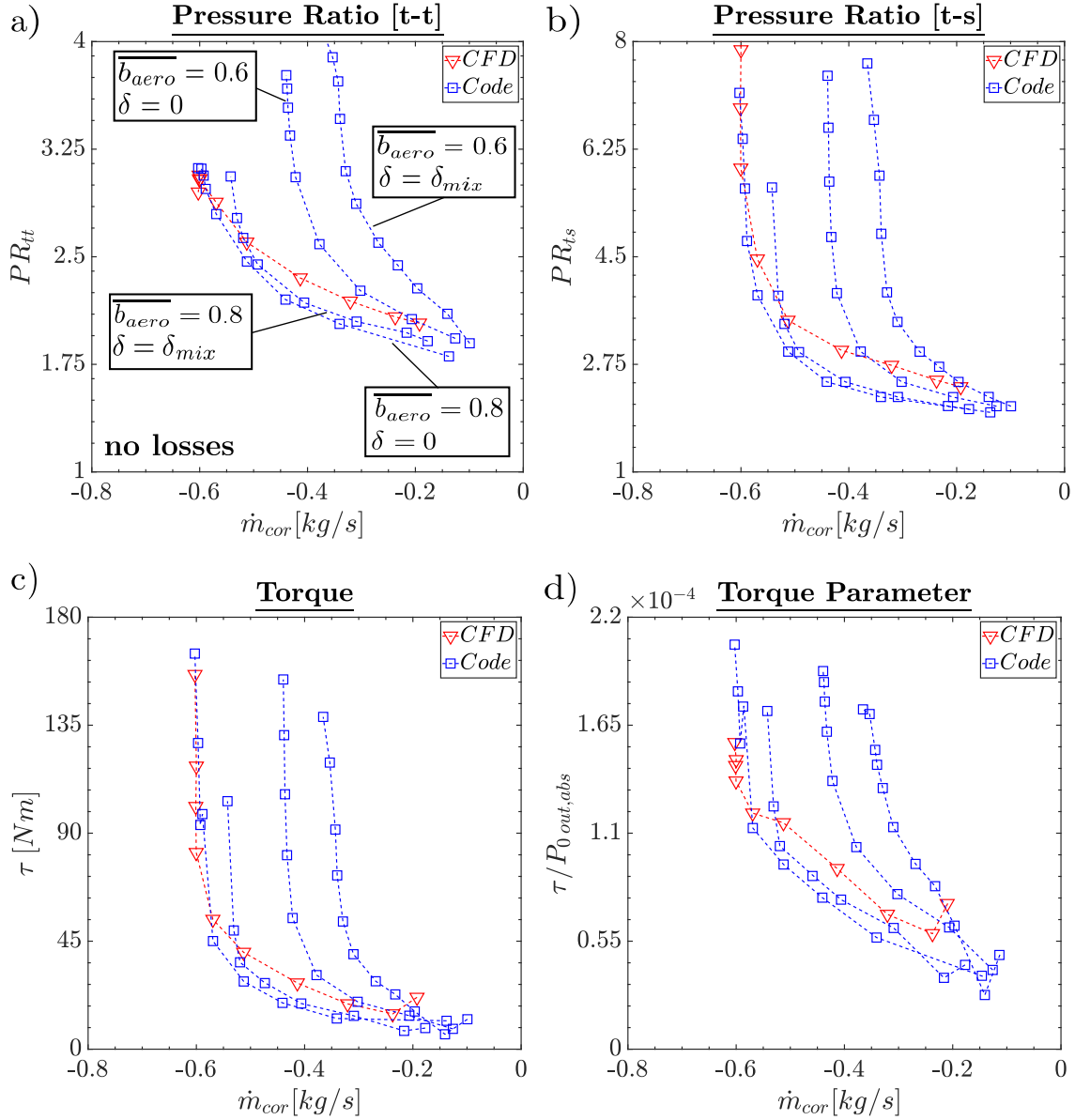


Figure 4.57: Effect of deviation due to mixing-out of the flow on the 60% reverse flow pressure and torque characteristics of impeller SRV2. Characteristics with and without deviation are compared at 20% and 40% aerodynamic blockage.

Loss effect

The impact of the losses inside the blade passage as well as downstream the duct is examined following a similar procedure. The "isentropic", reverse flow characteristic of the SRV2-0 centrifugal compressor at 80% speed is initially obtained, by aligning the turning force with the direction perpendicular to the actual relative path and not the blade. A blockage parameter corresponding to 20% blockage is imposed while deviation and additional mixing losses are omitted. Three additional characteristics are subsequently acquired by first imposing flow deviation, then deviation and mixing

losses at the duct and finally deviation, mixing losses downstream the blade and losses via the turning force model. The reverse flow characteristic obtained by the CFD model is also included for reference.

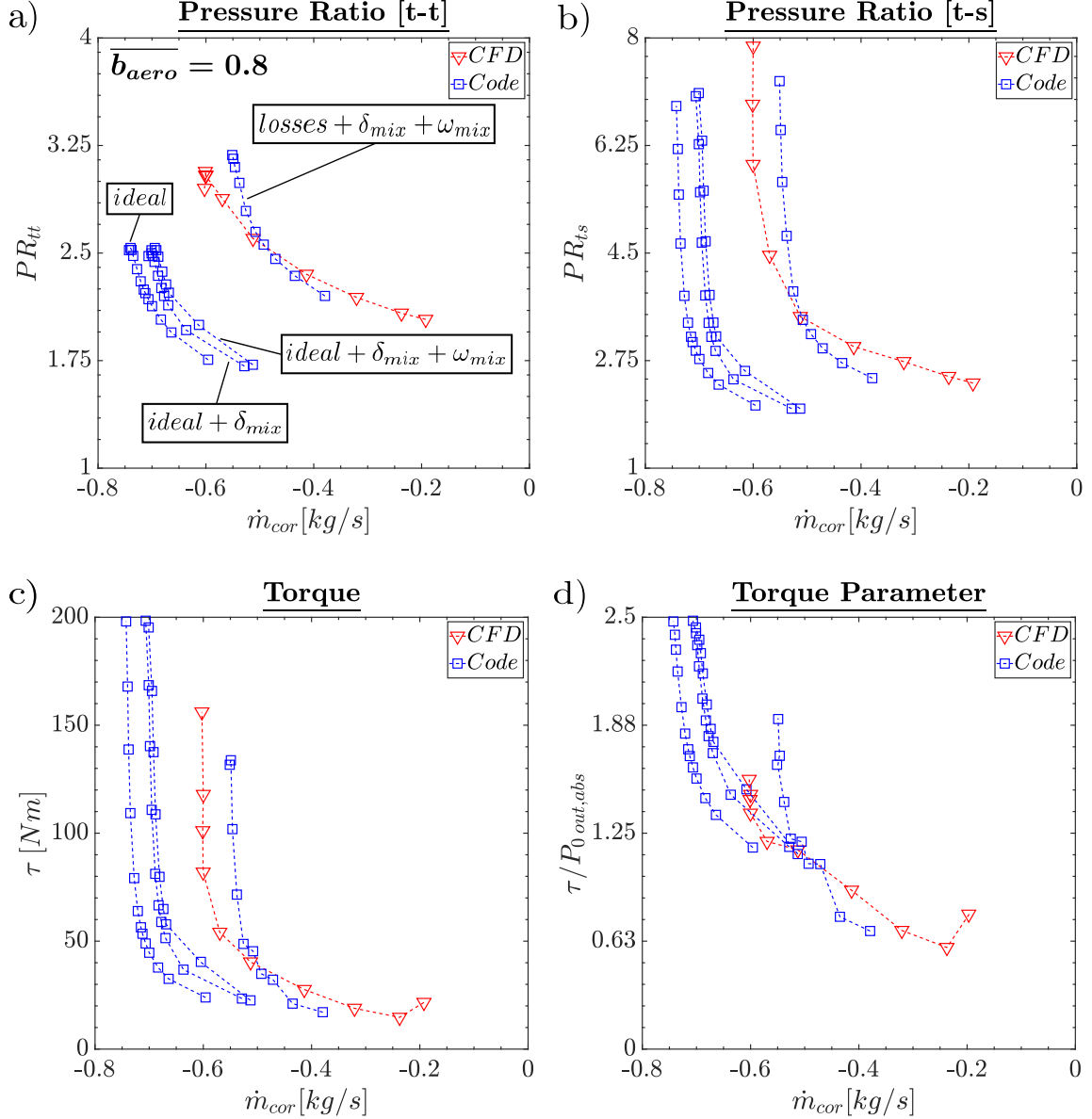


Figure 4.58: Effect of passage and additional mixing losses on the 80% reverse flow pressure and torque characteristics of impeller SRV2 with 20% aerodynamic blockage. The position of the characteristic due to the addition of losses and deviation is shown relative to the ideal one.

The characteristics obtained with 20% aerodynamic blockage and the different modelling approaches are depicted in figure(4.58). The "isentropic" characteristic is the least accurate one, significantly overestimating the mass flow capacity, while underestimating the pressure ratios. Imposing deviation and mixing losses moves the characteristics towards the lower mass flow region. These losses are of second order

compared to the losses introduced via the turning force inside the blade and their impact is notably less significant compared to that of flow deviation. It is evident that the losses introduced via the turning force model accompanied by a change in the relative flow angle due to flow separation are the most essential reverse flow modelling aspects since they dramatically change the position of the characteristic, shifting it upwards and towards the lower mass flow region. The intensity of those losses along with the flow turning can be modified by changing the extent of the modified curvature region. The implications of the transition region in the compressor reverse flow aerodynamic performance is discussed in the next section.

Modified curvature effect

The extent of the modified curvature region essentially dictates the extent of the flow misalignment in the blade passage which in turn affects the losses introduced via the turning force and the mass flow capacity of the compressor. Three characteristics at 40% design speed are obtained by introducing losses via the turning force and assuming zero aerodynamic blockage and therefore negligible flow deviation and duct mixing losses. Three characteristics are obtained by varying the extent of the transition region from 10%, then to 30% and finally to 60%.

The modified curvature region extending from the TE to the transition point, changes the imposed flow angle distribution as shown in figure(4.59), with the red solid line denoting the actual blade angle distribution at midspan. The mass flow averaged, rotary stagnation pressure loss distributions, corresponding to different modified curvatures are shown in figure(4.59b,c,d). In all three configurations the rotary stagnation pressure losses increase (and the mass flow) at higher pressure ratio settings. To investigate the effect of the modified curvature region in the losses, curves of similar pressure ratio need to be compared. Comparing the loss distributions of the 10% and 60% configurations corresponding to a pressure ratio of 1.41 and 1.65 designates an increase in the losses for the larger modified curvature configuration, by approximately 10% and 8% in the losses respectively. Except for the increased losses, the high-loss region near the TE is substantially larger and the imposed pressure losses are more gradually introduced.

The pressure ratio and torque characteristics are compared against the corresponding CFD-derived ones and are presented in figure(4.60). Within the modified curvature region, the flow is not turned isentropically thus higher losses are introduced the further it is extended. The modified curvature also results in increased mass flow capacity. The latter is evident by the shift of the pressure and torque characteristics

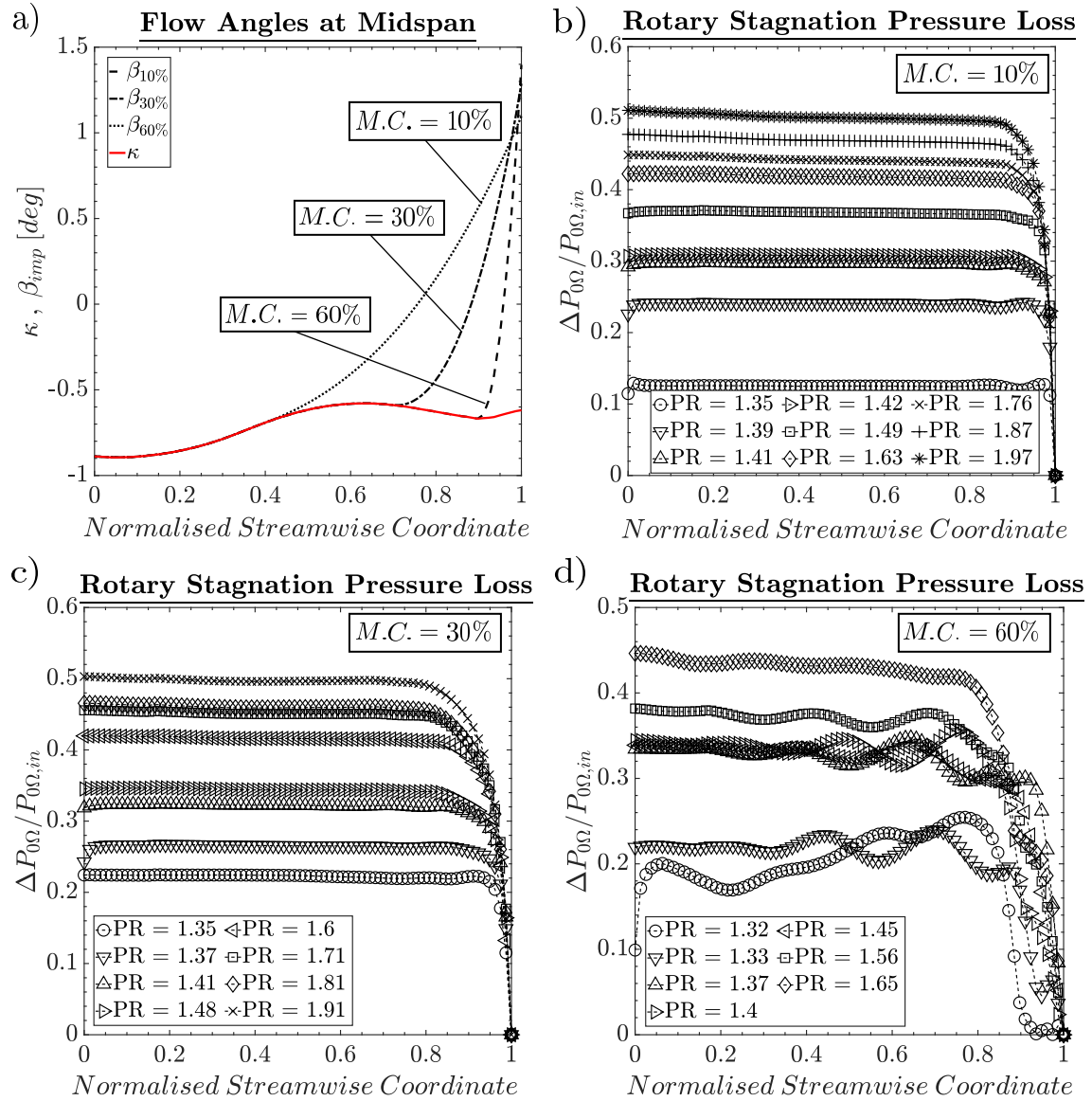


Figure 4.59: a) Imposed flow angles corresponding to different modified curvature lengths and streamwise distribution of mass flow-averaged rotary stagnation pressure losses in impeller SRV2 passage for b) 10% c) 30% and d) 60% modified curvature.

towards the lower mass flow region. Changes in the pressure ratio are not as pronounced, since the increase in losses of up to 10% have a rather second-order effect compared to the mass-flow capacity changes which is the dominant implication of the modified curvature region. While for impeller SRV2-0 a 10% modified curvature length was found to yield accurate results for both forward and reverse simulations, a 40% modified curvature was found more appropriate for the simulation of the NASA CC3 compressor with both vaneless and vaned diffuser at reverse flow conditions. This inevitably means that at reverse flow, the length of the modified curvature region exhibits tuning parameter characteristics. In essence, this parameter expresses

how rapidly the flow aligns with the blade, thereby conveying information about a physical phenomenon. Due to the large pitch length of impellers in the vicinity of the TE, values close to 40% are expected to be more realistic in impeller and vaned diffuser blades, while values closer to 10% in axial rotors and stators.

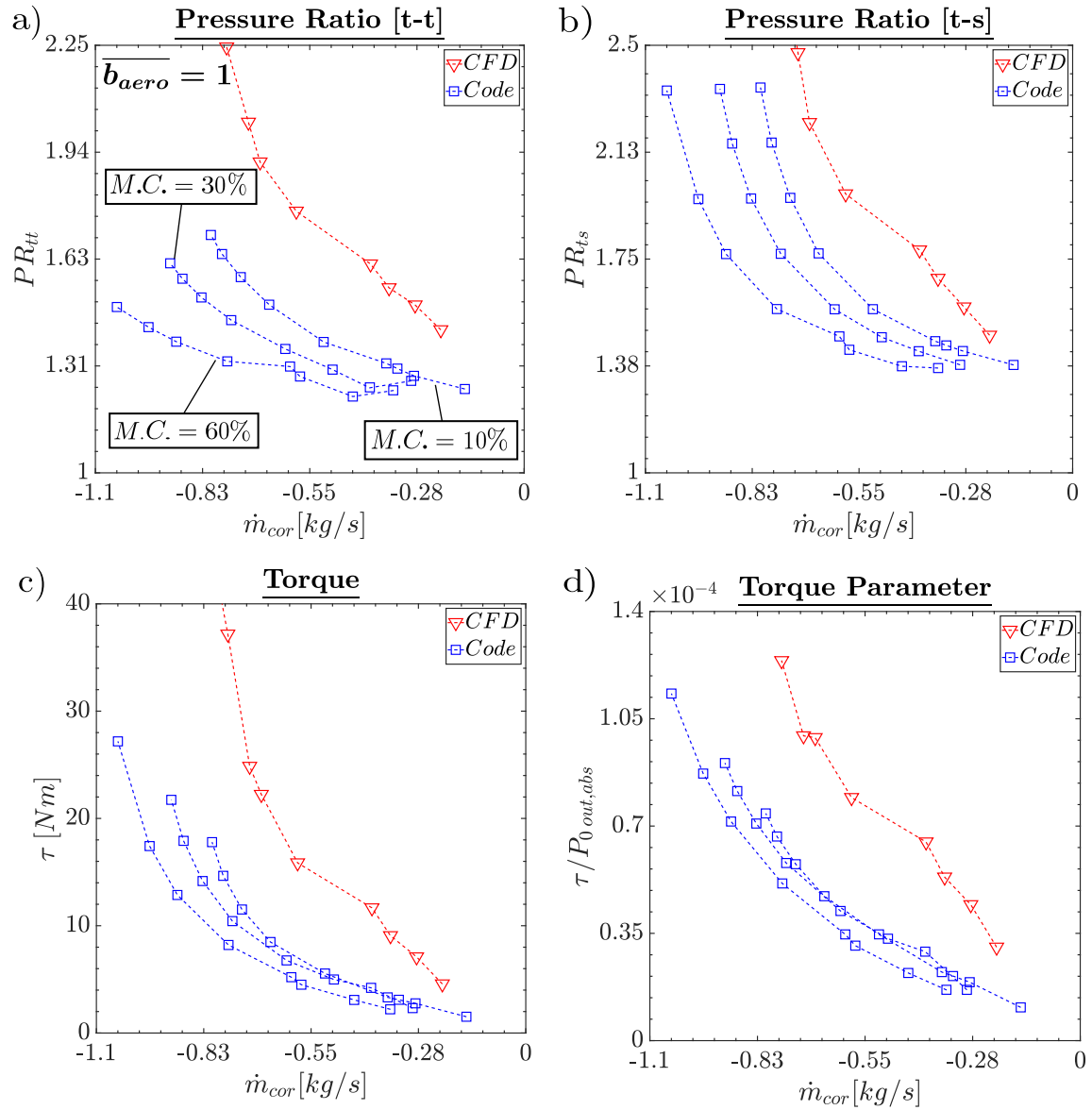


Figure 4.60: Effect of modified curvature length on the 40% reverse flow pressure and torque characteristics of impeller SRV2. aerodynamic blockage, mixing losses and deviation are neglected.

4.8 Chapter summary

In this chapter, the validation of the developed methodology is presented. The solver is verified in three steps: The correct implementation of the Riemann solvers is validated against the analytical solution of Sod shock-type of problems and the accuracy of three approximate and one exact Riemann solvers are compared. The HLLC and Roe's Riemann solvers are more accurate than the HLL and only marginally less accurate than the exact one. The HLLC is computationally more efficient than Roe's. Axisymmetric, empty duct simulations are performed and compared against the flow-field solution obtained with FLUENT. Numerical response tests verify the correct implementation of flux correction and the validity of the new blockage gradient evaluation method. Numerically-induced pressure losses of up to 4.8% due to blockage modelling are identified and are addressed using smoothening methods. The turning force model is validated against 4 compressor blade geometries including an axial fan rotor and stator, an impeller blade and a vaned diffuser blade. At the TE, the errors do not exceed 2% in any of the tested blade geometries. Unphysical, oscillatory behaviour due excessively high correction forces are discussed. Subsidiary losses due flow turning are reported but are found to be within reasonable limits. The viscous force model is validated in an axial rotor (rotor 37) and an impeller blade (SRV2). A prescribed range of average pressure losses is imposed and is compared against the average measured one. Errors higher than 8% are recorded when excessively high, unrealistic losses are imposed, while the errors are below 5% in the mid-to-low range which is representative of typical losses in axial and impeller blades. The distribution coefficient concept is extensively investigated using rotor 37 and CSTAR impeller geometry. A prescribed range of average pressure losses and deviation angles are imposed and the resulting, measured averaged quantities, derived from through-flow simulations are compared against the imposed ones. Simulations are performed mainly in rotor 37, using a combination of constant, CFD-derived and mixed distribution coefficients. The imposed quantities are in excellent agreement with the measured ones in both configurations employing constant and CFD coefficients. Pressure loss and flow angle errors do not exceed 4% and 1.4% respectively. Pressure and efficiency characteristics are extracted at 60% speed using constant, mixed and CFD coefficients with the characteristics exhibiting minor differences, confirming the validity of the distribution coefficient concept. Spanwise distributions are compared against CFD ones, revealing the beneficial effect of CFD coefficients for the reproduction of more physically-accurate flow-fields as well as limitations of the extraction and

implementation method. These are primarily related to underestimation of losses at the hub and the shroud which are presumably related to the assumptions of the viscous force extraction method and the implementation approach of the coefficients in the viscous force model. The curvature definition method is validated against numerical and experimental data available in the public domain. A preliminary verification of the reverse flow modelling method is presented. The aerodynamic blockage predictions of the new model are compared against predictions of 2 other models found in the literature. The Powers et al. model is substantially more sensitive to incidence angle variations, predicting larger variations with respect to the other 2 models. The predictions of the new model are in good agreement with Longley's. Unlike axial compressor blades, significant mixing-out of the jet and wake flow structure occurs inside the impeller and vaned diffuser passage. The assumption of the analytical model of negligible deviation angle at LE is confirmed by CFD data. Larger deviation angles manifest predominantly as a result of other flow phenomena in the vicinity of the blade passage, e.g. recirculation regions and vortical structures. The losses due to flow separation and other mechanisms inside the passage are in good agreement with CFD data, but are consistently overpredicted. Losses due to mixing of the flow downstream bladed regions, is in good agreement with CFD results but the presence of large recirculation regions did not allow for the extraction of reliable pressure loss and flow angle data. The effect of losses, flow deviation, aerodynamic blockage and modified curvature on global aerodynamic reverse flow performance is assessed. Blade passage loss is the most dominant reverse flow modelling aspect, significantly affecting both pressure ratio and mass flow capacity. Additional mixing losses inside ducts are of second order, while aerodynamic blockage substantially alters the shape of the characteristic and the mass flow capacity of the compressor. Flow deviation increases the energy exchange, while decreasing the choking mass flow. Larger modified curvature regions induce up to 10% higher losses, while substantially increasing the choking mass flow, shifting the characteristics towards the lower pressure ratio and higher mass flow region.

Chapter 5

Pre and post-stall aerodynamic performance

This chapter elaborates on the steady-state and transient aerodynamic response of different compressor geometries at forward and reverse flow conditions. The validation process at steady-state, forward flow conditions is carried out in three phases. Initially, global aerodynamic performance quantities (pressure ratio, temperature ratio and efficiencies) are compared against experimental and CFD data in a range of rotational speeds. Spanwise profiles of the aforementioned quantities downstream the compressor are subsequently compared against the corresponding CFD-derived distributions. The flow-field solution acquired by the through-flow code is finally assessed qualitatively, by comparing the derived contours of certain flow-field variables, against the corresponding, CFD ones. The steady-state, reverse flow verification process of the code is based exclusively on CFD data due to absence of experimental data. Reverse flow characteristics of total-to-total and total-to-static pressure ratio as well as torque and torque parameter are compared against characteristics derived from CFD simulations. Visualisation of the flow-field provides a further qualitative assessment of the through-flow code's solution, unveiling deficiencies of the artificial shear stress terms. The reverse flow verification is focused exclusively on centrifugal compressors, in particular impeller SRV2 and NASA CC3 with vaneless and vaned diffuser. Transient post-stall simulations are performed with the through-flow code using the 1D-plenum throttle model. The post-stall response of the compressor is investigated for different plenum volumes and the accuracy of the predicted surge cycle trajectory is evaluated using steady-state, forward and reverse flow characteristics. The chapter is concluded by validating surge on NASA CC3 compressor with vaned diffuser. Static pressure measurements at the vaned diffuser throat during a surge event consisting of 3 cycles

are obtained by the code and compared against the experimental data, successfully predicting the frequency, the magnitude and the dynamic response of the compressor.

Contents

5.1 Forward flow validation	246
5.2 Reverse flow	293
5.3 Chapter summary	322

5.1 Forward flow validation

The verification process of the solver, the body force model and the reverse flow modelling discussed in the previous section, confirmed the ability of the developed modelling approach to accurately reproduce essential, turbomachinery-related flow features e.g. flow turning, losses, shock waves, blockage etc. Certain limitations associated with their implementation were identified, with the introduction of subsidiary losses due to blockage, flow turning and over or underprediction of viscous forces being the most salient ones. These errors were shown to be within a very reasonable range, thus their effect is expected to be minor and of second order in comparison to errors stemming from the limitations and accuracy of the empirical loss correlations which provide estimations of flow deviation, pressure loss and aerodynamic blockage at forward flow conditions. These quantities are predominantly dictating global compressor aerodynamic performance, while the accurate reproduction of the flow-field is mostly dependent upon the overall modelling approach. The accurate reproduction of the flow-field is inevitably subject to the limitations of the solver since complex, viscous-related aerodynamic 2D as well as 3D phenomena, for instance flow separation, boundary layers, vortices, mixing etc. cannot naturally be reproduced by solving the axisymmetric inviscid Euler equations. Through-flow simulations are performed in 5 different blade geometries. These geometries were briefly presented in section(4.2) and comprise NASA rotor 37 (axial compressor rotor), NASA stage 67 (axial fan stage), SRV2-0 compressor (larger centrifugal compressor with vaneless diffuser), NASA CC3 compressor (larger centrifugal compressor with vaned diffuser) and CSTAR compressor(smaller centrifugal compressor with vaned diffuser).

5.1.1 NASA rotor 37 - Forward flow

To assess the accuracy of the code, rotor 37 is first modelled using the code as well as a CFD model. The obtained pressure and efficiency characteristics are subsequently compared against experimental data. Spanwise distributions and flow-field contours of certain quantities obtained by the code are also compared against CFD data to evaluate the capability of the code to reproduce essential flow features. A visualization of the 3D geometry generated by the code is shown in figure(5.1).

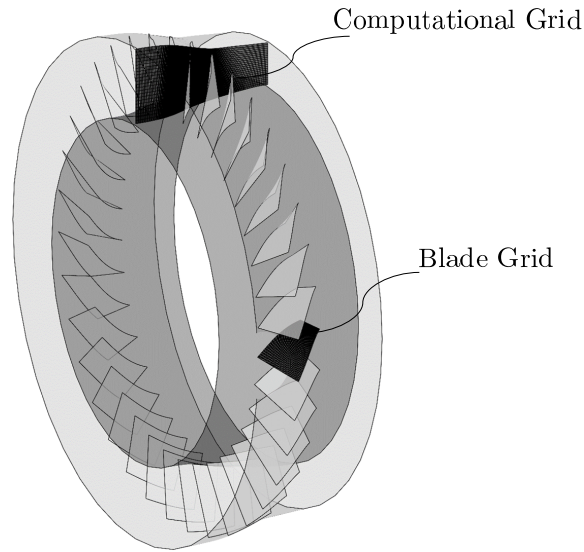


Figure 5.1: 3D visualization of axial compressor NASA rotor 37. The main computational duct grid along with the camber surface grid is also shown.

Figure(5.2) shows a single passage mesh of rotor 37 generated in ANSYS Turbogrid. The grid is structured following guidelines of a previously validated mesh [84] and comprises approximately 1 million elements, achieving y^+ values well-below 11. The flow-field solution is obtained using RANS simulations; the $\kappa - \omega$ *SST* turbulence model is used, while the solution of the advection and turbulence terms is performed using second order schemes. Since no stationary blade rows are present in the model, all domains are modelled as rotating domains, with angular velocity identical to the one of the rotor, thus bypassing the need for mixing models.

Figure(5.3) shows the pressure, suction and camber surface grids of rotor 37 as well as the meridional projection of the camber surface mesh used to define the blockage and the curvature of the blade. Due to the aspect ratio of the blade, a larger number of spanwise elements is used to obtain accurate blade curvature estimations. A relatively dense mesh is preferred in the streamwise direction, to ensure blade curvature estimations with sufficiently high resolution at the LE and TE.

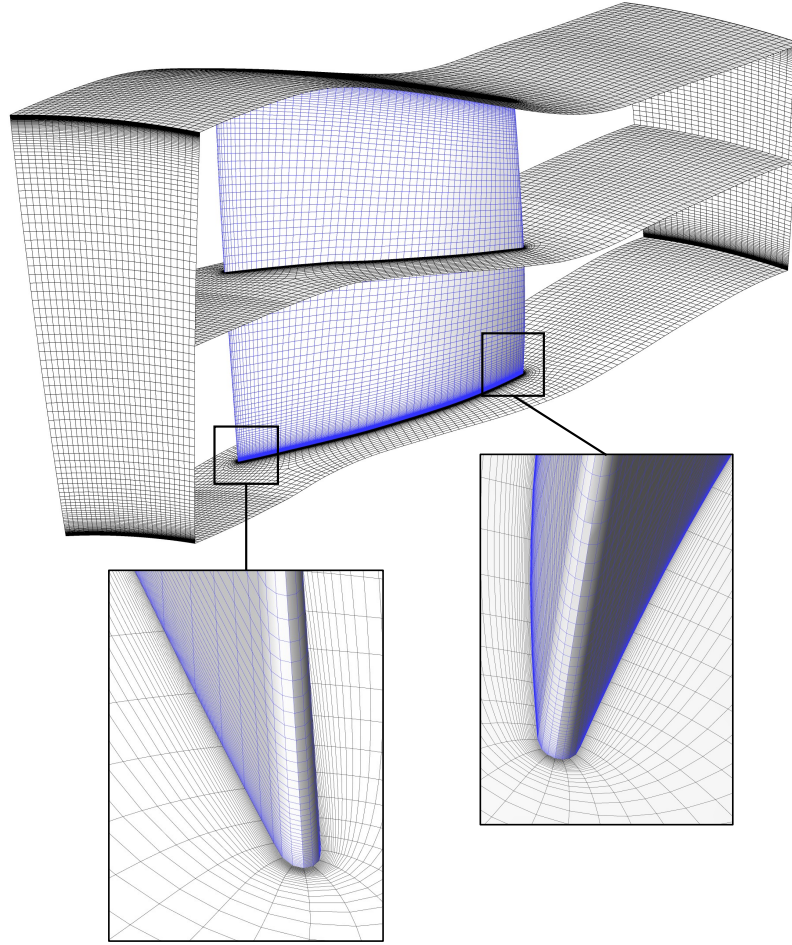


Figure 5.2: Overview of rotor 37 computational domain used for single-passage RANS simulations.

The flow-field solution is obtained on the computational grid, shown in figure(5.4). The grid is generated using the adaptive approach, while transition regions with variable mesh distribution in the streamwise direction are adjoining the bladed domain with the coarser inlet and outlet domains to prevent numerical, mesh-related errors. The rotor subdomain is discretised with a significantly larger number of streamwise elements (110) compared to the 24 elements used in the radial direction. The large number of streamwise elements are necessary to gradually impose flow turning and prevent numerical instabilities. The mesh comprises in total approximately 12k elements. Uniform total pressure and temperature boundary conditions corresponding to standard ISA conditions are imposed at the inlet, while the static pressure at the outlet is varied to obtain different operating points on the characteristic. The distributions of the flow angles and the metal blockage are omitted since they were presented in detail in section(4.5).

The global aerodynamic performance predictions obtained by the code and the

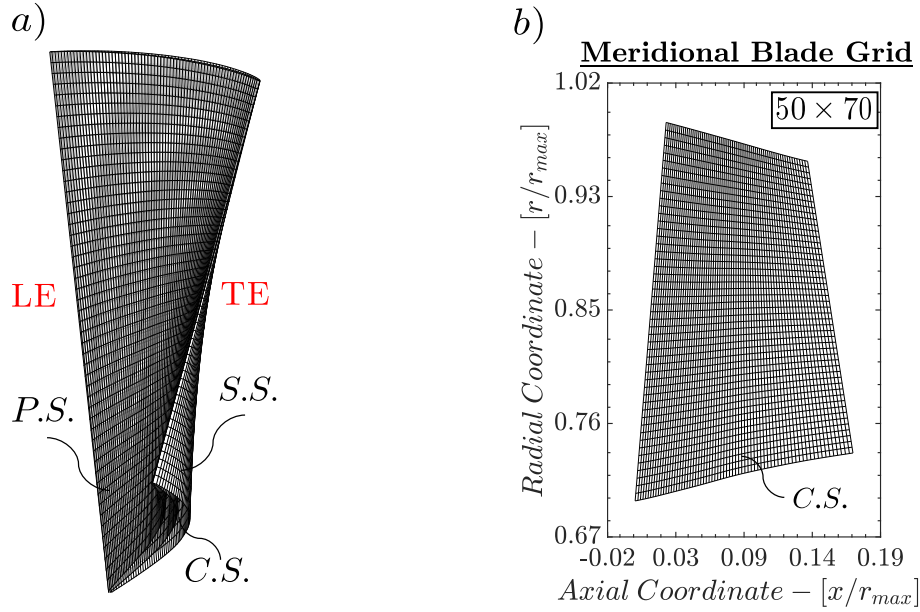


Figure 5.3: Overview of rotor 37 a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.

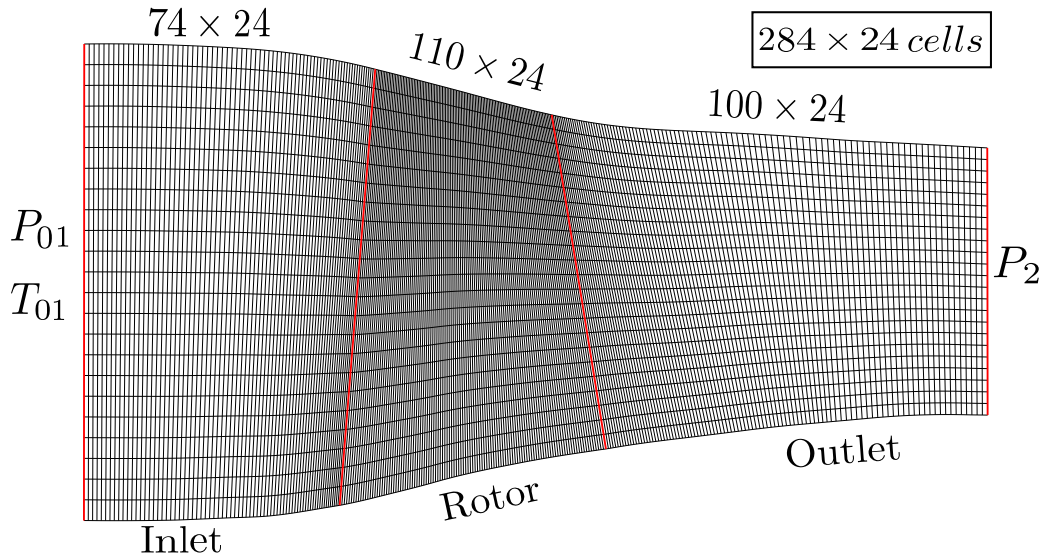


Figure 5.4: Adaptive computational domain of rotor 37 blade generated by the code to obtain the flow-field solution.

CFD model are presented in figure(5.5) and are compared against the experimental characteristics. The predictions of the CFD model in the lower speed range are in excellent agreement with the experimental results, while the pressure ratio at design speed is underpredicted by approximately 3%. The efficiency characteristics are slightly underpredicted at all speeds by up to 4.5% , 7.5% and 4.3% at 60%, 80% and 100% design speed respectively. The CFD characteristics indicate higher flow turning, due to less flow deviation and higher losses, resulting in accurate pressure ratio but

lower efficiency predictions. The pressure ratio estimations obtained by the code are consistently overpredicted, however the shape of the characteristic is consistent with the trends of the experimental results at all design speeds. The pressure rise predictions are in good agreement with the experimental ones but tend to be less accurate at lower speeds with the errors being in the range $[5.5\% - 8.2\%]$ at 60% design speed, while the corresponding errors do not exceed 4.7% and 4.3% at the 80% and 100% design speed respectively. Larger errors, up to 12% are recorded in the efficiency characteristics, with the errors being more pronounced in the higher mass flow region. It is evident that these errors manifest as a result of the erroneous estimations of the empirical loss and deviation models which substantially overpredict the losses, while underpredicting flow deviation. This leads to high pressure rise and low efficiencies.

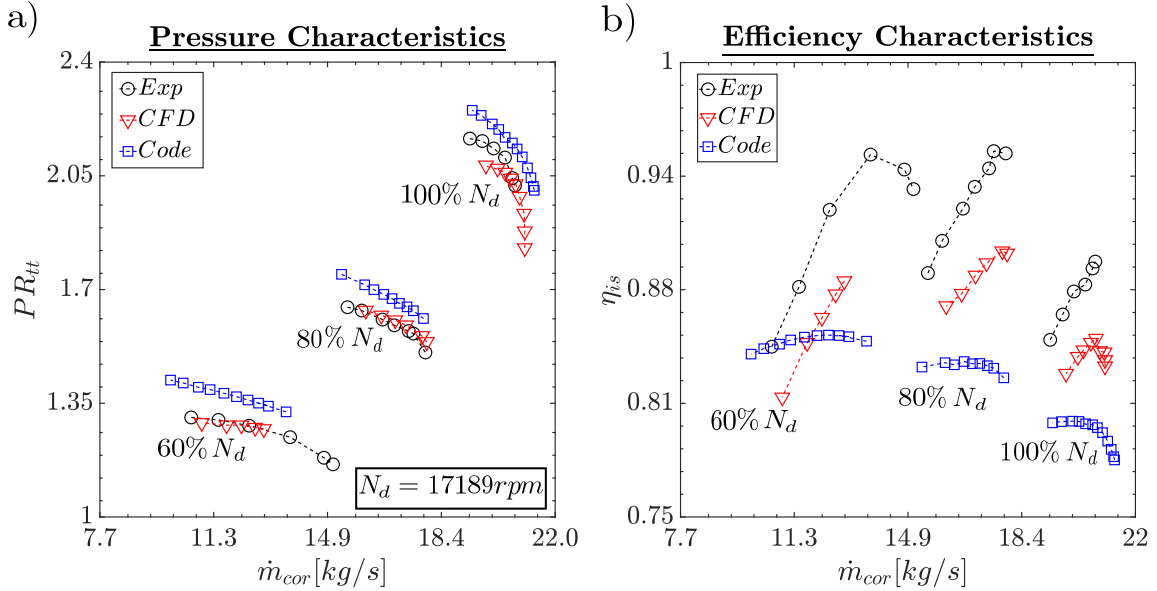


Figure 5.5: Comparison of NASA rotor 37 experimental pressure ratio and efficiency characteristics against CFD and code predictions using empirical models coefficients proposed by Aungier.

The through-flow predictions shown in figure(5.5), were obtained by setting the profile loss coefficients K_1 and K_2 according to Aungier's suggestion, as per eqn.(3.218). Tuning of those coefficients and the coefficients determining the design deviation angle, K_{sh} and $K_{t,\delta}$ can drastically alter both the precision as well as the shape of the characteristics. Figure(5.6) depicts the pressure and efficiency characteristics obtained after readjusting the tuning coefficients of both the deviation and the loss empirical model. These readjustments resulted in substantial improvements in efficiency predictions but introduced a detrimental effect regarding the shape and

accuracy of the pressure ratio characteristics at the 80% and 100% design speeds. Further readjustment of the coefficients was not attempted since it is a limitation of the empirical models and is considered out of the scope of the present work. Definition of those coefficients on a spanwise basis and potentially as function of flow-field variables is expected to yield substantial accuracy improvements.

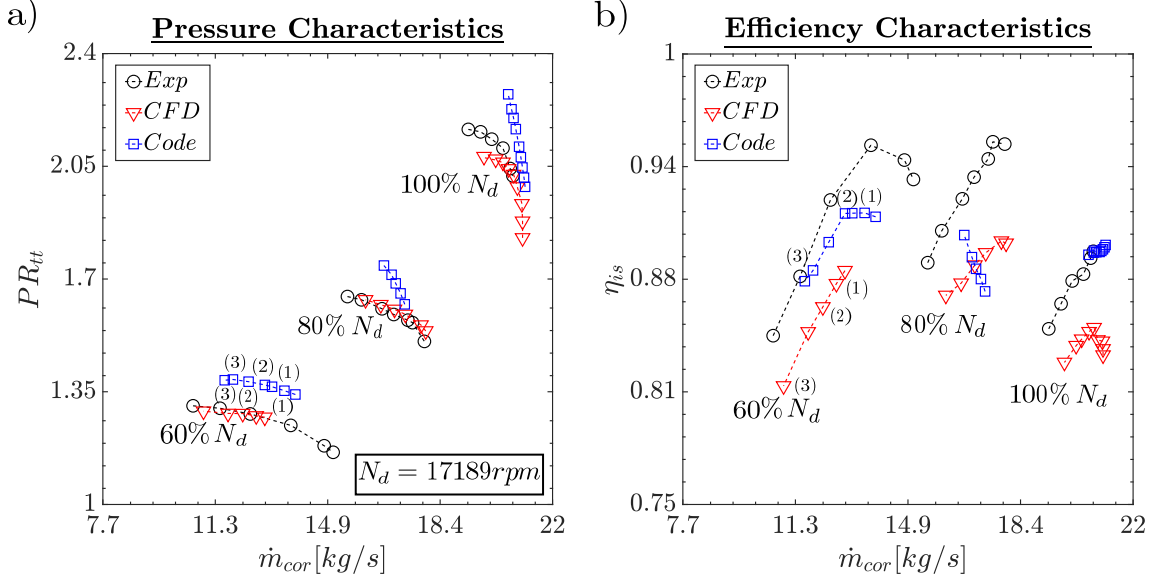


Figure 5.6: Comparison of NASA rotor 37 experimental pressure ratio and efficiency characteristics against CFD and code predictions using different set of empirical models coefficients.

To investigate the ability of the code to realistically reproduce essential flow-features, spanwise distributions of pressure ratio, temperature ratio and efficiencies of the retuned 60% design speed characteristic are extracted just downstream the TE (5.7). Due to mismatch of the predicted pressure and temperature ratios, the quantities are normalised with their corresponding peak values. A single distribution coefficient, derived from the design speed characteristic, is used to obtain all the operating points along the three characteristics shown in both figures (5.5) and (5.6). The coefficient is obtained in the higher mass-flow ratio region and thus is more representative of point (1) on the 60% characteristic. The spanwise distributions of the aforementioned quantities correspond to three operating points as indicated on figure(5.6).

The pressure ratio distribution corresponding to point(1) successfully captures the reduction in pressure ratio near the hub and the shroud due to boundary layer and tip leakage vortex effects, although this effect is slightly more pronounced in the code. The temperature ratio trends are in good agreement with the CFD ones, with the code successfully reproducing the increased power input due to the presence of

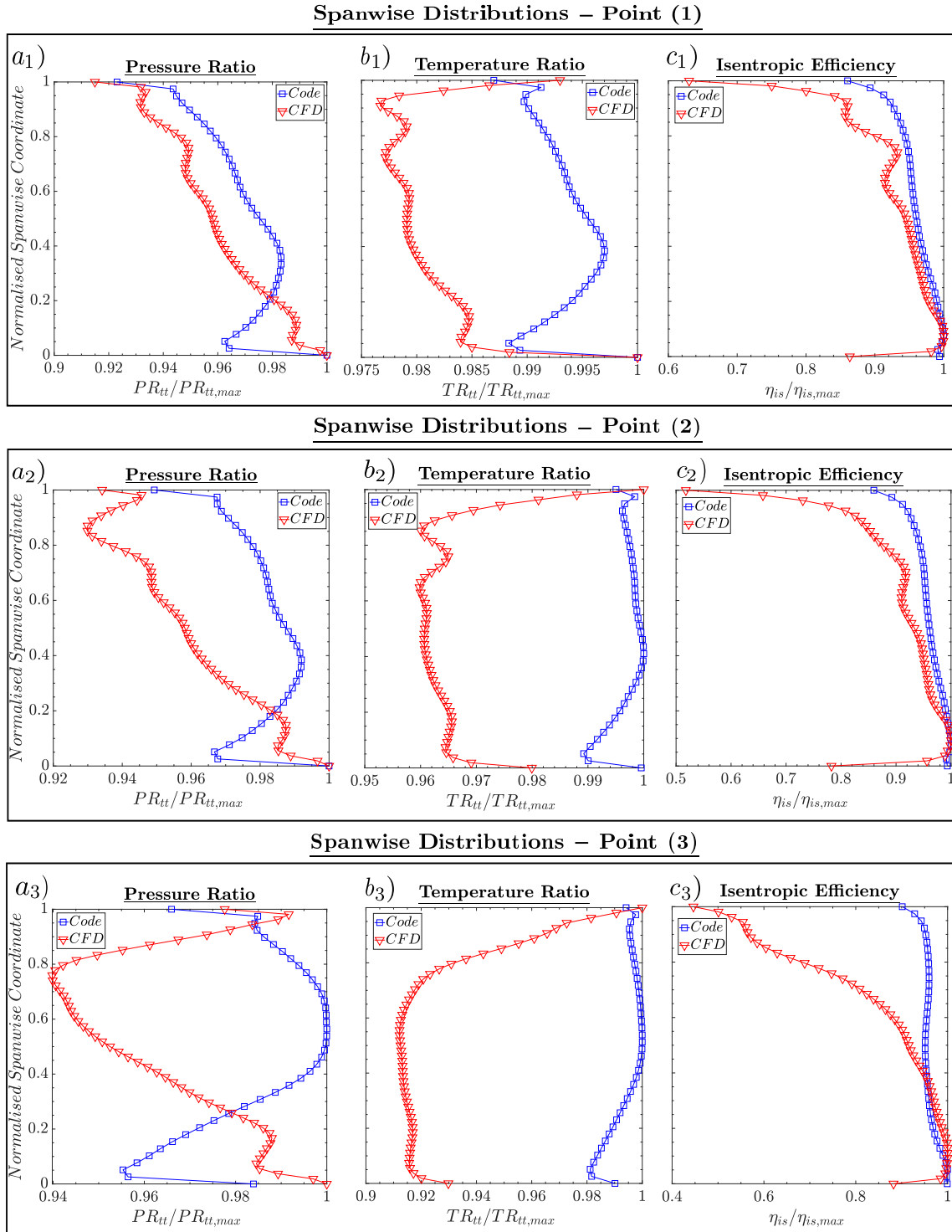


Figure 5.7: Efficiency, total pressure and temperature ratio spanwise profiles downstream the TE of rotor 37 obtained by through-flow and CFD simulations at three different operating points on the 60% characteristic.

larger viscous forces at both the hub and the shroud. At the shroud, the presence of the tip leakage vortex results in higher losses and therefore increased power input to achieve the same flow turning (the flow does not separate). The intensity of the

losses is underestimated by the distribution coefficient, which is also evident by the higher efficiency predictions of the code (figure(5.7 c_1)). It should be noted that the pressure and temperature ratio variations in (figure(5.7 a_1, b_1)) are rather insignificant since they correspond to less than 4% and 2.5% respectively. Notable errors in the efficiency distributions can be seen in the vicinity of the hub and the shroud where the viscous losses are patently underpredicted by the code. The distributions corresponding to the second operating point exhibit similar trends and are captured reasonably well, however the errors are slightly exacerbated due to the increased intensity of tip leakage vortex which further offsets the maximum values with which the curves are normalised. This is much more pronounced in operating point (3) which corresponds to near-stall conditions, where it is evident, that the excessive intensity of the viscous-related phenomena, primarily the tip leakage vortex and the separated flow at the shroud and the boundary layer at the hub, cannot be properly captured using the same distribution coefficient.

The distribution of the flow angles, shown in figure(5.8) is accurately reproduced in all operating points. The most notable differences are the offset of the curves, indicating the underestimation of flow deviation by the deviation correlations and the change in the flow angles at the hub which is not captured by the code. A contour of the flow angle distribution corresponding to the higher mass flow operating point is depicted in figure(5.9). The through-flow distributions indicate the imposition of consistently increased flow turning, apart from a small region near the hub, reflected on the lower flow angles at the TE.

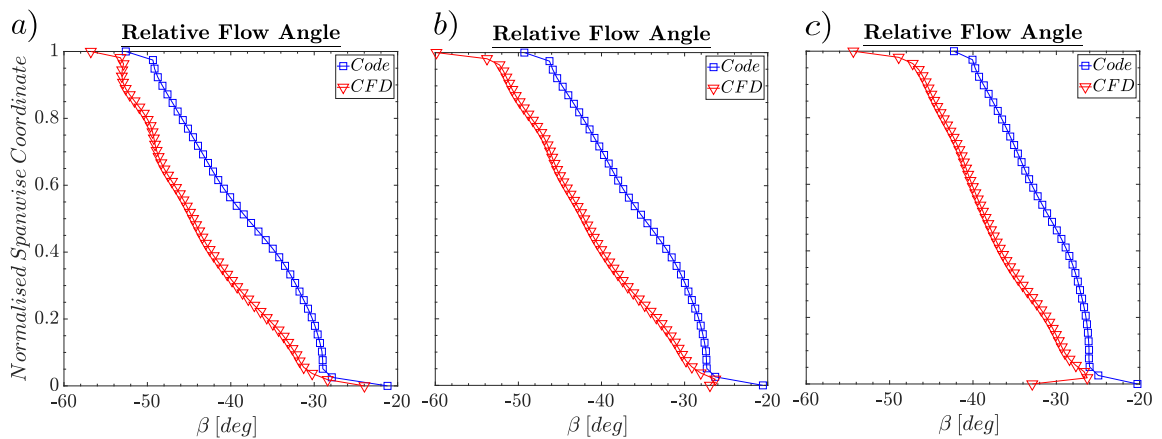


Figure 5.8: Comparison of spanwise flow angle distributions obtained by through-flow and CFD simulations at 3 operating points on the 60% characteristic of rotor 37.

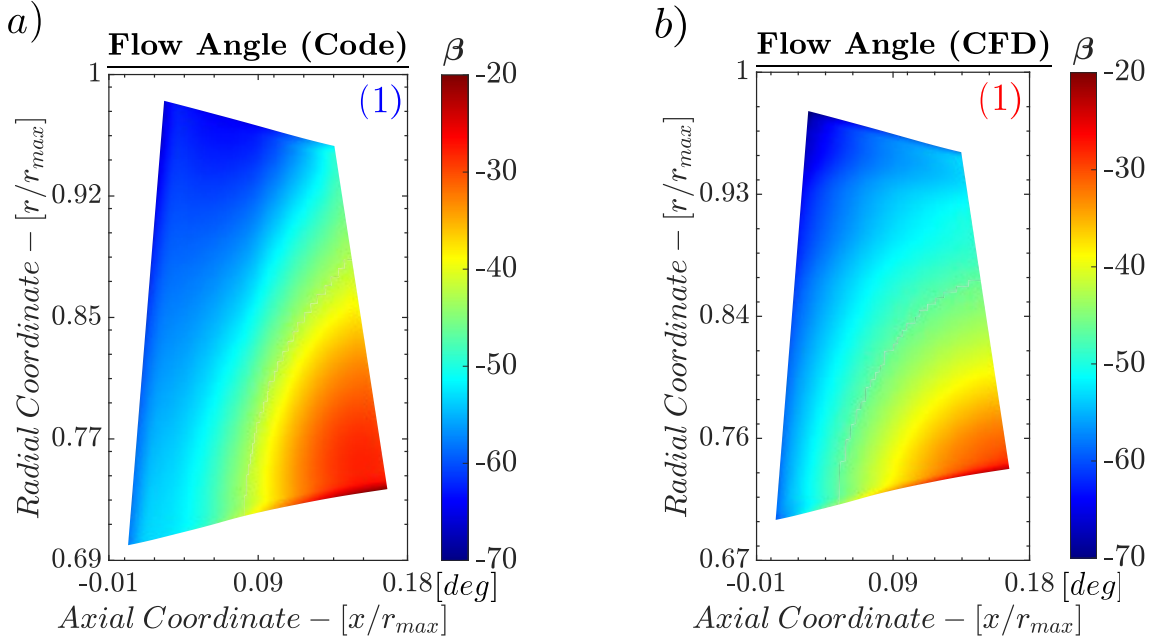


Figure 5.9: Flow angle contours at a high mass flow operating point at 60% speed obtained from a) the code and b) CFD simulations.

Flow-field contours of total pressure and temperature as well as contours of static pressure and meridional Mach numbers are subsequently compared against CFD contours (figures(5.10) and (5.11)). The through-flow contours are in good agreement with the CFD ones, with the most notable differences being a high static pressure - low velocity region near the LE, towards the shroud (figures (5.10a) and (5.11a)) and a small total pressure drop region at the shroud, downstream the TE. The first phenomenon manifests due to the modified blade curvature at the inlet, while the second manifests due to flow separation at the TE. The transition region modifies locally the curvature of the blade, temporarily changing the direction of the turning force, thus decelerating the flow and increasing the static pressure. Downstream the transition region, the flow is aligned with the blade, thereby following the actual imposed curvature and changing the direction of the turning force; this results in a rapid acceleration of the flow. This effect is more pronounced near the shroud due to the larger LE angles and turning forces and is a corollary of the modified curvature approach used to ensure numerical stability. This phenomenon can be suppressed by reducing the extent of the transition region (here set to 10%) although this could potentially give rise to numeric instabilities. The static pressure contours are displayed with a slightly different colorbar limits to better highlight the pressure distribution downstream the TE which is accurately reproduced by the through-flow code.

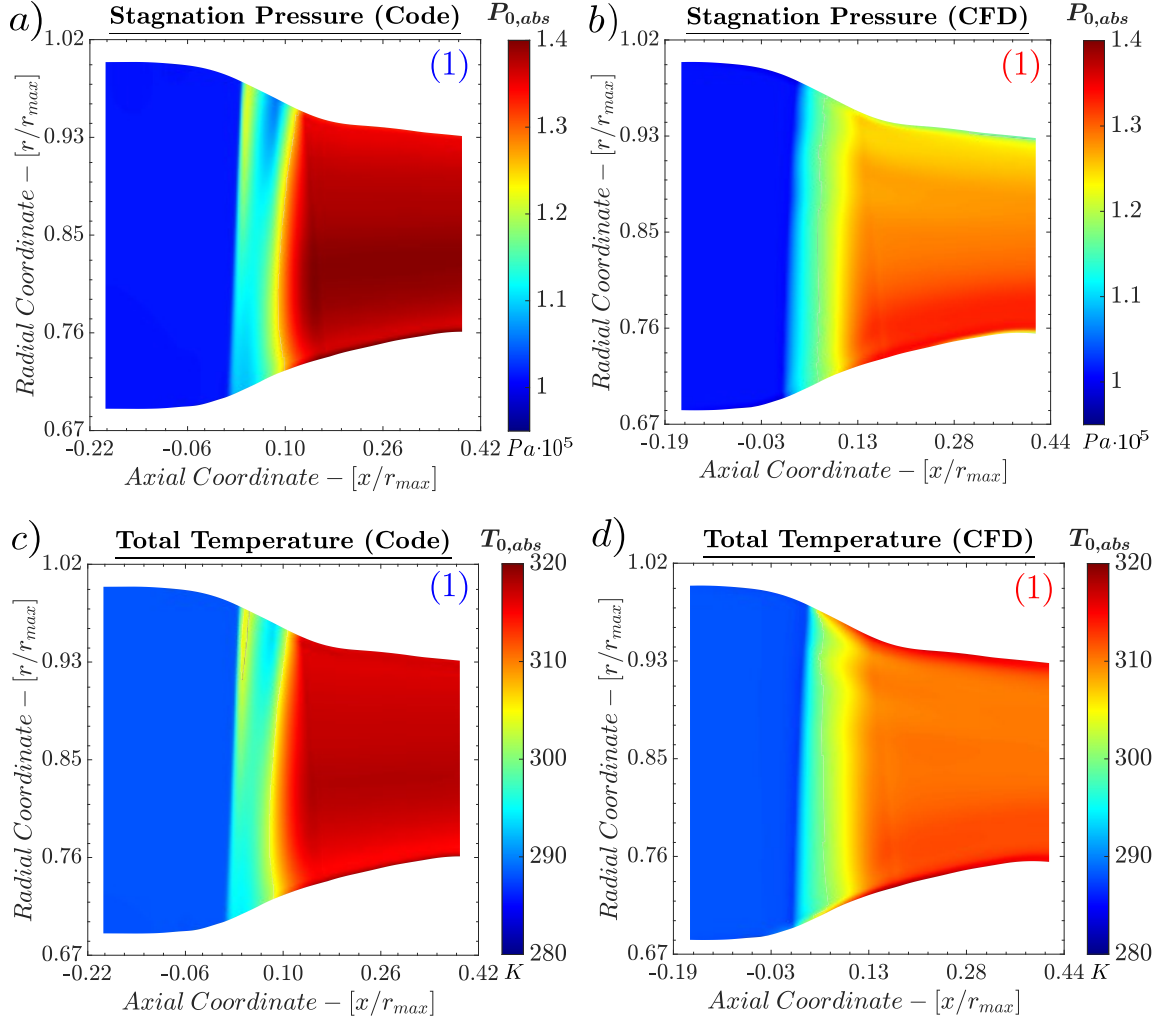


Figure 5.10: Comparison of through-flow against CFD total pressure and temperature contours of rotor 37, at a high mass-flow operating point on the 60% characteristic.

The flow separation near the TE, evident by the reduction in total pressure and low velocity of the CFD solution (figures(5.10b) and (5.11b)) is a complex, viscous phenomenon and cannot be recreated by the through-flow code, unless the locally imposed viscous force and flow deviation cause the flow to decelerate enough to induce flow separation (the turning force is a function of the local relative velocity). The separated flow is nevertheless confined within a small region near the TE and its contribution to the formation of the overall, steady-state flow-field is insignificant. Despite the aforementioned, inherent limitations the through-flow modelling method, the derived flow-field solution is in very good agreement with the CFD one, successfully capturing the essential features of the flow structure.

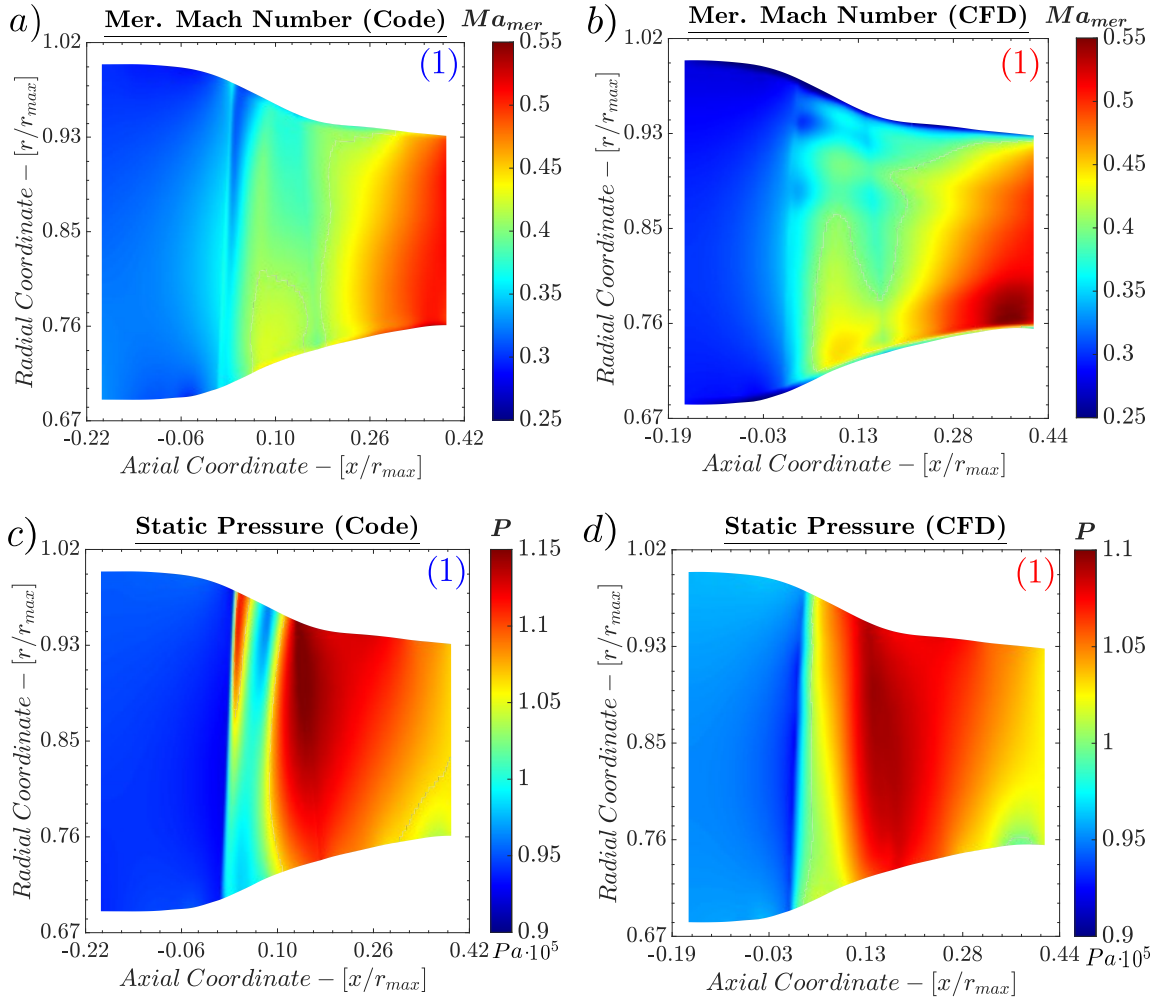


Figure 5.11: Comparison of through-flow against CFD meridional Mach number and static pressure contours of rotor 37, at a high mass-flow operating point on the 60% characteristic.

5.1.2 NASA stage 67 - Forward flow

The applicability of the overall modelling approach to axial compressor blades is further evaluated on NASA stage 67 (figure(5.12)), the first stage of a compact, two-stage fan configuration. Global aerodynamic performance predictions obtained by the code are compared against experimental pressure and efficiency characteristics. The geometry is not modelled in a commercial CFD software, therefore standard, uniform distributions coefficients are used to distribute both the losses and flow deviation. In section(4.4), global aerodynamic performance predictions were proven to be independent of the coefficients but the utilization of uniform distribution coefficients is expected to yield a less realistic flow-field since they are unable to reconstruct localised, viscous induced phenomena. The validation process is therefore limited to global aerodynamic performance predictions.

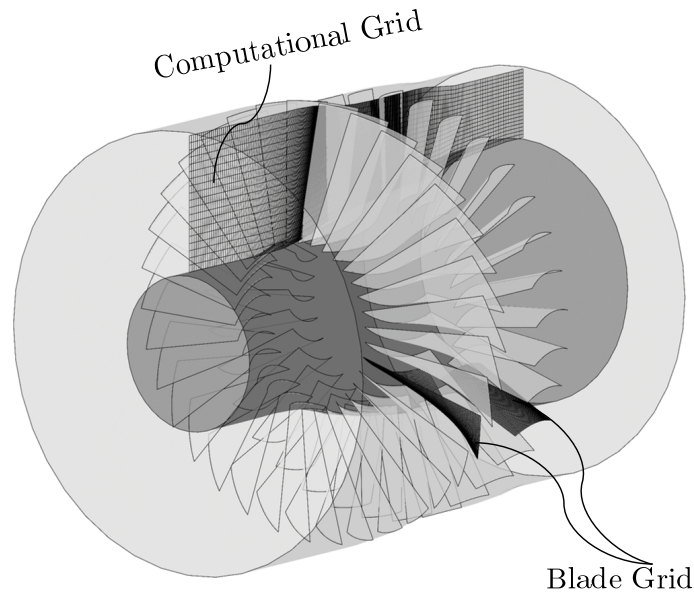


Figure 5.12: 3D visualization of axial fan stage NASA 67. The main computational duct grid along with the camber surface grid is also shown.

Figures (5.13) and (5.14) show the pressure, suction and camber side grids of rotor and stator 67 respectively as well as the meridional projection of the corresponding camber surface meshes on the meridional plane. To ensure a more uniform discretization, both blade grids comprise a larger number of elements in the spanwise direction. Both blade grids comprise 50 streamwise and 70 spanwise elements with a total element count of 350 elements for each blade surface.

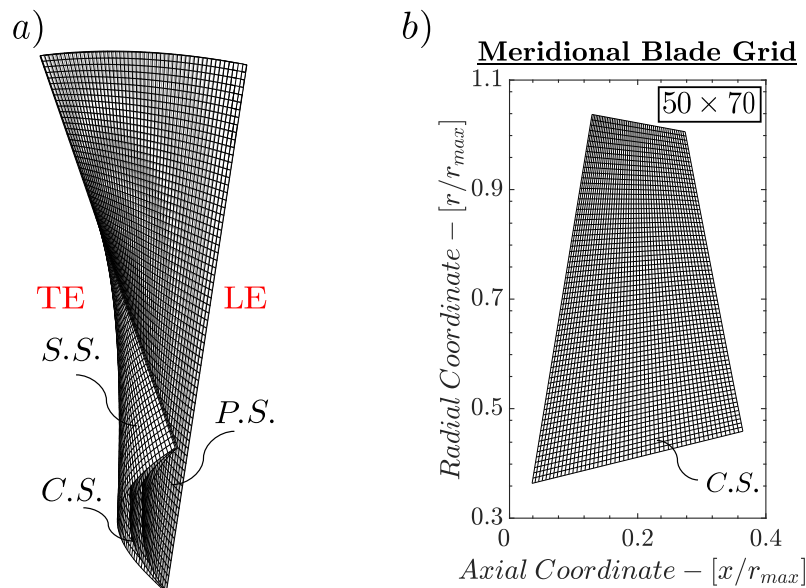


Figure 5.13: Overview of rotor 67 a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.

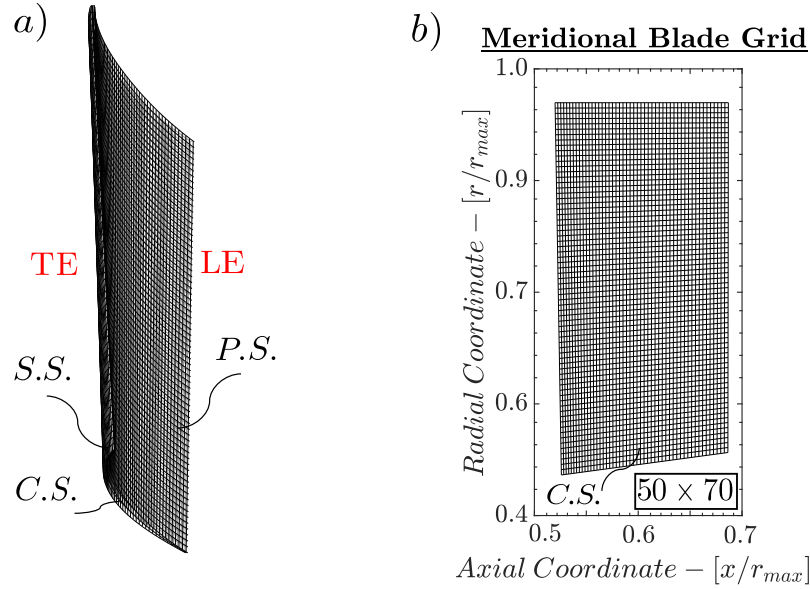


Figure 5.14: Overview of stator 67 a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.

The computational grid generated by the code is shown in figure(5.15). The grid is generated using the adaptive mesh approach, while transition regions are included inside subdomains located upstream and downstream the LE and TE of both the rotor and the stator. This enables the definition of subdomains with different grid resolutions while preventing numerical errors. The inlet and the outlet are modelled with a coarser grid, while a significantly denser mesh is used to discretise bladed regions. The rotor and the stator comprise 80 and 70 streamwise elements respectively. The grid consists of 8120 elements. Uniform, invariable total pressure and temperature boundary conditions (ISA) are imposed at the inlet, while the static pressure at the outlet is varied to obtain different points along the characteristics.

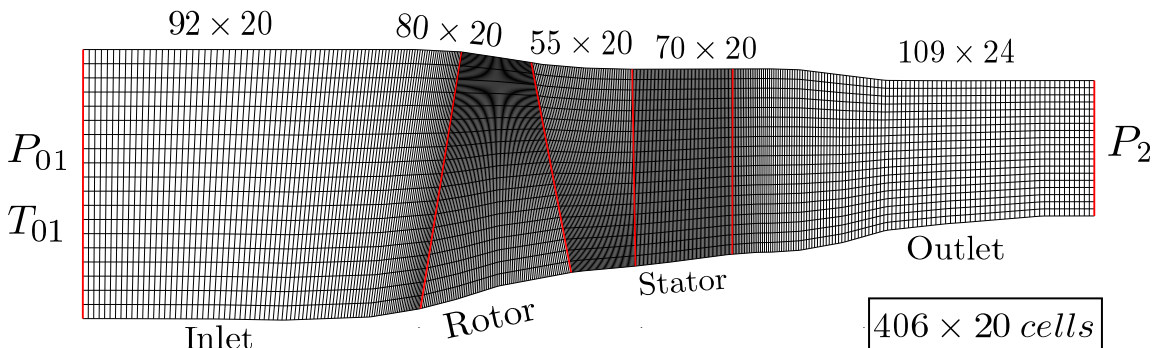


Figure 5.15: Adaptive computational domain of stage 67 blade generated by the code to obtain the flow-field solution.

The complete curvature definition of rotor 67 and stator 67, defined on the nodes of the camber surface grids depicted in figures(5.13) and (5.14) is shown in figures (5.16) and (5.17) respectively. The blade angle distribution is similar to the one of rotor 37, however the overall flow turning is substantially higher. Due to the larger curvature of the duct at the hub, flow path angles of up to 15 degrees are encountered, however they diminish in the midspan region. Blade leaning along the largest part of the blade is insignificant, however noteworthy blade leaning is observed near the TE, especially near the hub. Substantial reduction in the effective area of up to 30% can be seen near the root of the rotor blade.

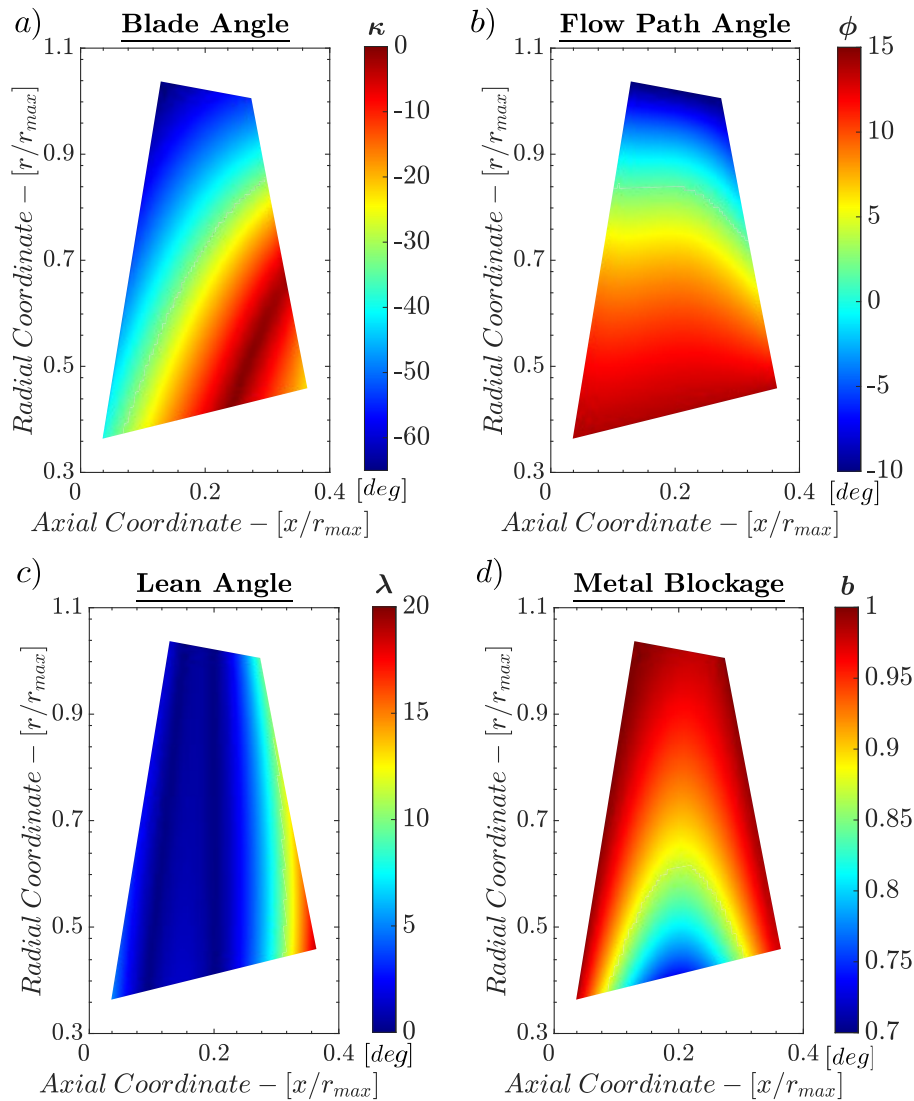


Figure 5.16: Blade angles and metal blockage parameter contours of rotor 67 blade.

The blade angle of stator 67 (figure(5.17a)) is very uniformly distributed in the spanwise direction, imposing large flow turning which results in negative blade angles at the TE. The effect of the flow path and the lean angle is rather negligible since

the curvature of the duct is aligned with the axial direction, while lean angles of less than 10 degrees are confined entirely within a small region in the LE close to the hub. Contrary to rotor blades, where the thickness of the blades diminishes substantially closer to the shroud, a uniformly distributed metal blockage in the spanwise direction of up to 15% can be observed in figure(5.17d).

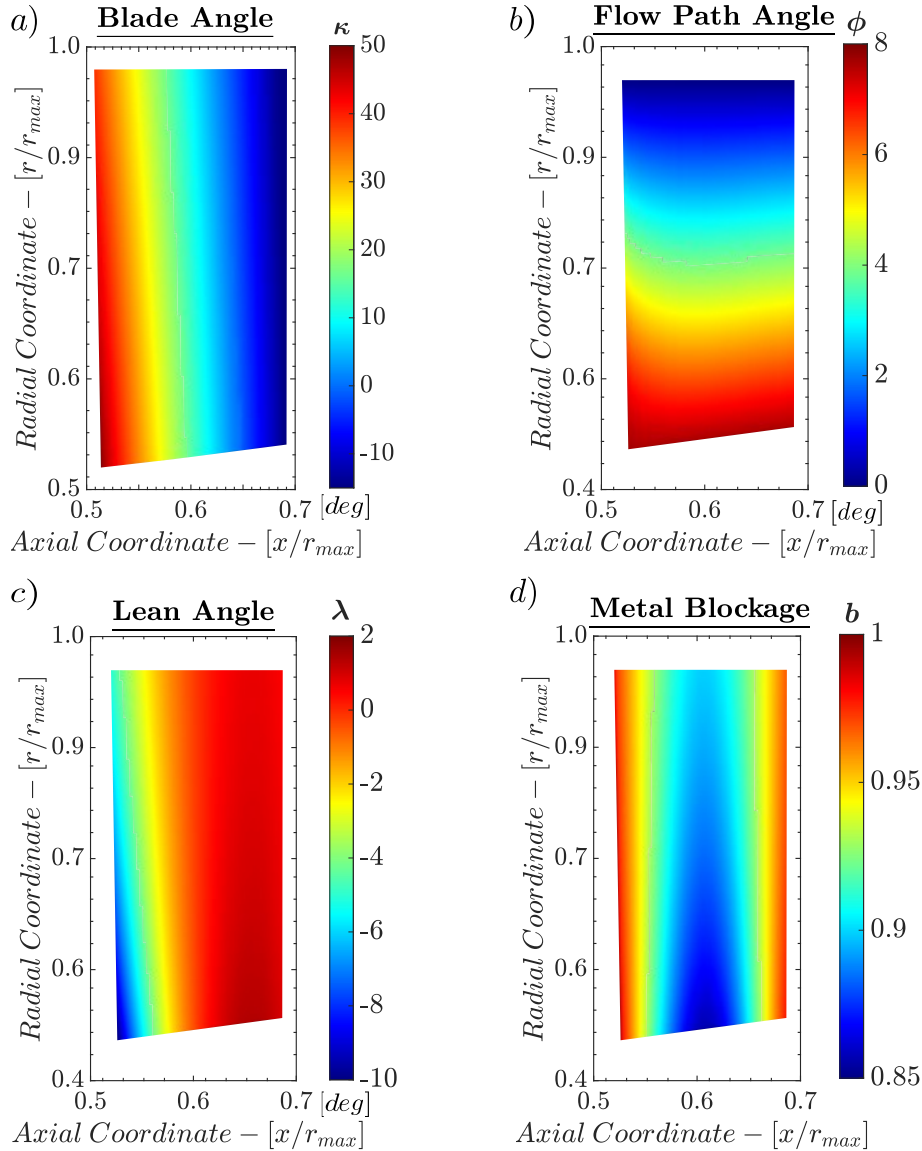


Figure 5.17: Blade angles and metal blockage parameter contours of stator 67 blade.

The pressure and efficiency characteristics are derived at 60%, 70%, 80%, 90% and 100% design speed and are subsequently compared against the experimental ones. Simulations are initially performed by setting the tuning parameters of both the loss and the deviation model following the guidelines of Aungier (figure(5.18)). The characteristics exhibit a similar behaviour to the ones of rotor 37 presented in the previous paragraphs; the pressure ratio characteristics are in good agreement

with the experimental data, with the pressure rise being consistently overpredicted, except for the higher design speed region, where the pressure ratio is underpredicted by up to 1.6%. The gradient of the pressure characteristics especially at design speed, is not consistent with the experimental one, primarily in the higher mass flow region. The efficiencies are substantially underpredicted, with the errors being in the order of 20%. The overestimation of pressure ratios is the outcome of the underpredicted flow deviation combined with the overestimation of the losses, with the first being the dominant source of error. The increased energy input due to higher flow turning is balancing out the excessive pressure losses, resulting in higher pressure ratios and a dramatic drop in the efficiency.

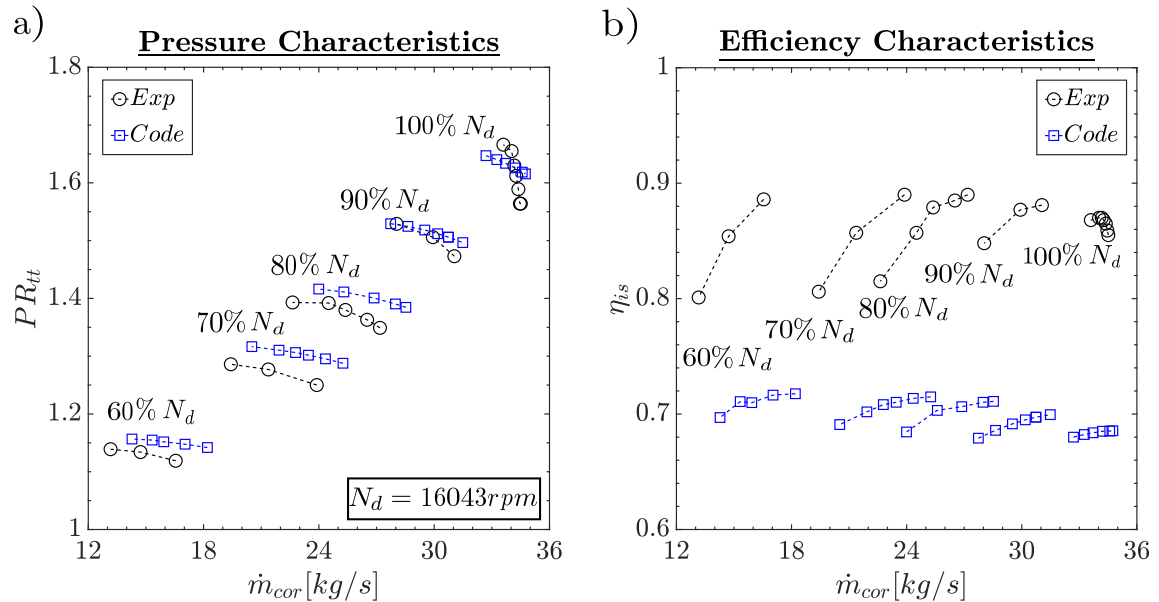


Figure 5.18: Comparison of NASA stage 67 experimental pressure ratio and efficiency characteristics against code predictions using empirical models coefficients proposed by Aungier.

Considering the aforementioned analysis, the sources of the mismatch stem from the overestimation of the pressure loss along with a concurrent underestimation of the flow deviation. To address the sources of the error, the empirical loss coefficients are readjusted accordingly and the aerodynamic performance predictions obtained with the updated configuration are compared against the experimental data (figure(5.19)). The pressure ratio predictions as well as the gradient of the characteristics are in very good agreement with the experimental results, particularly in the 80% and 90% range, where the errors are less than 2.2% and 1% respectively. The gradient of the predicted pressure characteristic at design speed is notably less steep, while the maximum pressure ratio and the corresponding mass flow is underestimated by

1.35% and 4.47% respectively. The efficiency characteristics correlate remarkably well with the experimental data, primarily in the lower speed characteristics and the lower mass flow range. The peak efficiency at design speed is underpredicted by approximately 2.5%, while the errors in the lower design range do not exceed 3.1%. This test case further highlights the high dependency of axial compressor aerodynamic performance predictions on the precision of the empirical correlations which in turn, are highly sensitive to the shape/correction coefficients of those models. The extensive validation of the solver and the body force model presented in chapter(4) provided insight into subsidiary losses, misalignment due to flow turning and errors in the imposed losses. The uncertainty originating from those errors was concluded to be within a very reasonable range and is further verified by the results which designate the loss correlations as the primary source of error.

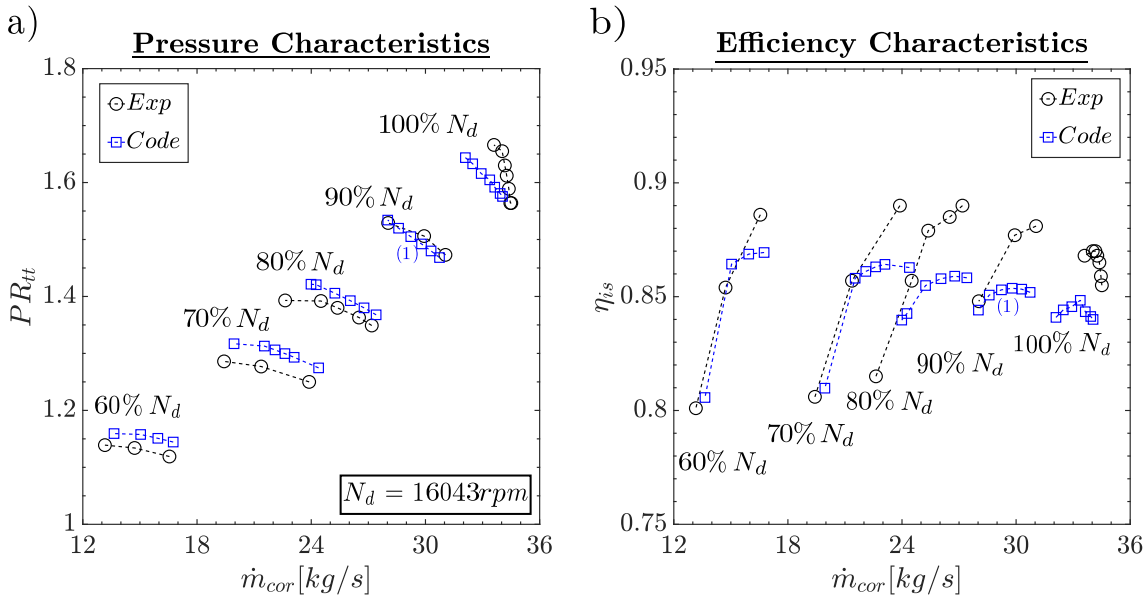


Figure 5.19: Comparison of NASA stage 67 experimental pressure ratio and efficiency characteristics against code predictions using different set of coefficients for the empirical models.

5.1.3 DLR SRV2-0 - Forward flow

To verify the applicability of the developed code to centrifugal compressors, the centrifugal compressor SRV2-0 is modelled in the code, using a similar approach to the one presented in the previous section. A CFD model is subsequently developed and the results of the numerical models are compared against experimental data to evaluate the accuracy of global aerodynamic performance predictions. The ability of the code to reproduce essential flow-features is quantitatively and qualitatively assessed

by comparing spanwise distributions and flow-field contours of certain quantities of interest at different operating conditions. A visualization of the 3D geometry is shown in figure(5.20). The compressor geometry comprises an inlet bell-mouth, an impeller and a vaneless diffuser.

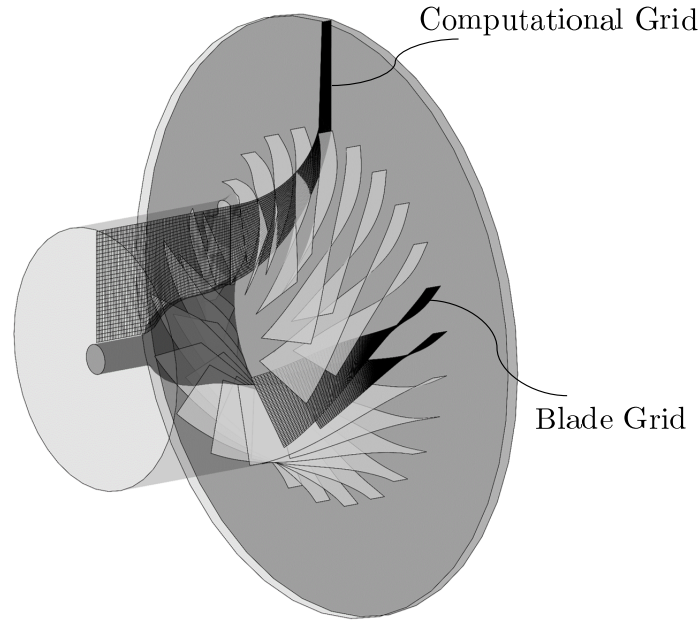


Figure 5.20: 3D visualization of SRV2 centrifugal compressor with vaneless diffuser. The main computational duct grid along with camber surface grid is also shown.

Figure(5.21) depicts a single passage mesh of impeller SRV2 generated using ANSYS Turbogrid. The blade passage incorporates both a full and a splitter blade as well as the corresponding circumferential sector of the inlet bellmouth geometry and the vaneless diffuser. The grid comprises approximately 3.6 million elements, ensuring that y^+ values are not exceeding 11, an essential requirement for the wall functions. Based on the outcomes of GCI analyses by [122] and [123], a grid of approximately 1 million elements is sufficient. This was verified by comparing the dense grid (3.6 million elements) against a coarser one, consisting of 1.3 million elements, showing negligible variations in pressure ratio and efficiencies. Nevertheless, all CFD results were obtained with the denser grid. The steady-state solution (RANS) of the flow-field is performed in ANSYS CFX solver using the $\kappa - \omega SST$ turbulence model and second order-accurate schemes for the advection and turbulence terms. Due to the absence of stationary bladed domains, all domains are modelled as rotating ones, thereby avoiding the need for mixing planes.

The curvature of the blade is defined along the camber surface mesh of the full blade, shown in figure(5.22a). Since splitter blade geometries are essentially trimmed versions of the full blades, they exhibit identical curvature distributions. While the

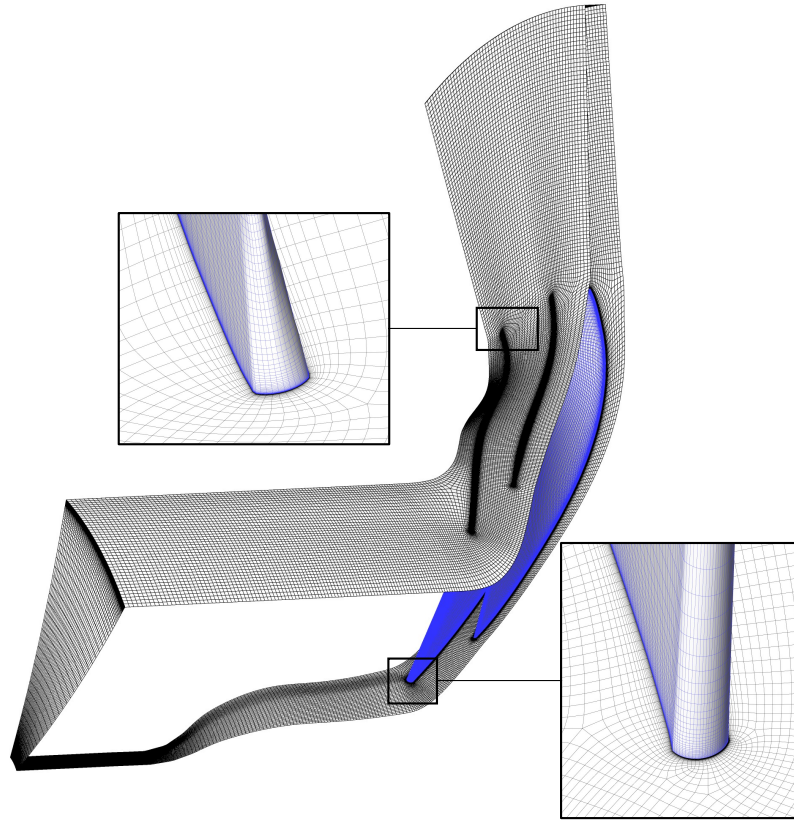


Figure 5.21: Overview of SRV2-0 compressor computational domain with vaneless diffuser used for single-pass RANS simulations.

presence of the splitter blades is not explicitly interfering in the turning of the flow due to the formulation of the body force model which assumes that the flow is guided by an infinite number of blades, their presence is accounted for in a two-fold way; increase in the metal blockage and change in the effective number of blades, an intrinsic quantity affecting both the empirical loss and deviation model predictions. The blockage of the splitter blade is defined along a distinct splitter blade grid, generated via interpolation based on the 3D coordinates of the splitter blade. Due to the long aspect ratio of impellers, the blades are discretised using a larger number of elements in the streamwise direction, which is essential for accurate curvature estimation near the LE and the TE.

Figure(5.23) shows the computational grid generated by the code following the adaptive mesh generation method. Due to the more complex geometry of the inlet bell mouth as well as the large change in the radial position and consequently the cell volumes and face areas in the diffuser domain, a denser, more uniformly structured grid was deemed essential. A lower resolution grid in the vaneless diffuser space could potentially precipitate numerical errors, due to sudden changes in the cell face areas and volumes of adjacent cells. This is highly exacerbated due to the large changes in

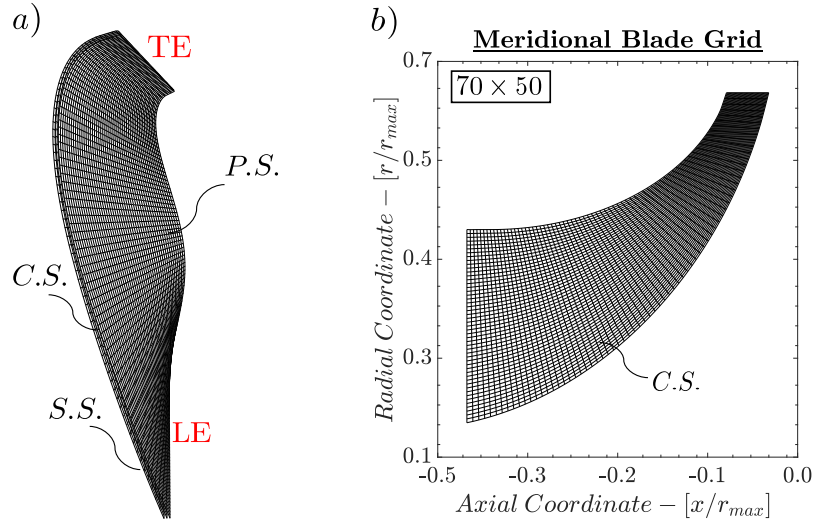


Figure 5.22: Overview of impeller SRV2 a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.

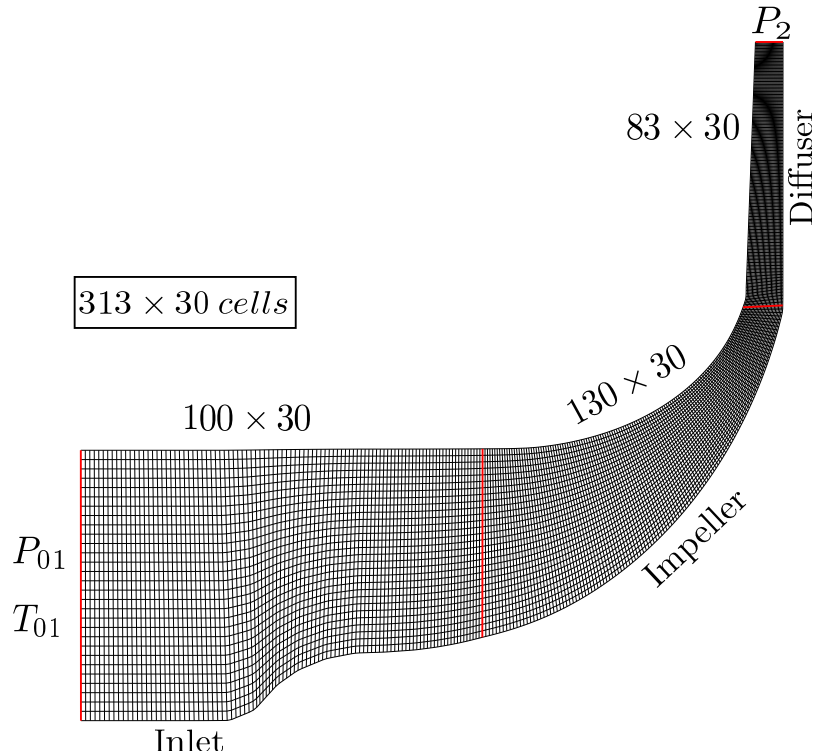


Figure 5.23: Adaptive computational domain of impeller SRV2 generated by the code to obtain the flow-field solution.

the radius and can result in unrealistic flow phenomena similar to the ones discussed in section(4.1.3). The grid consists of 3 subdomains, corresponding to the inlet bell mouth, impeller and diffuser, with the bladed domain comprising 130 elements in the streamwise direction. Transition regions are not added, while the total number of elements is 9390.

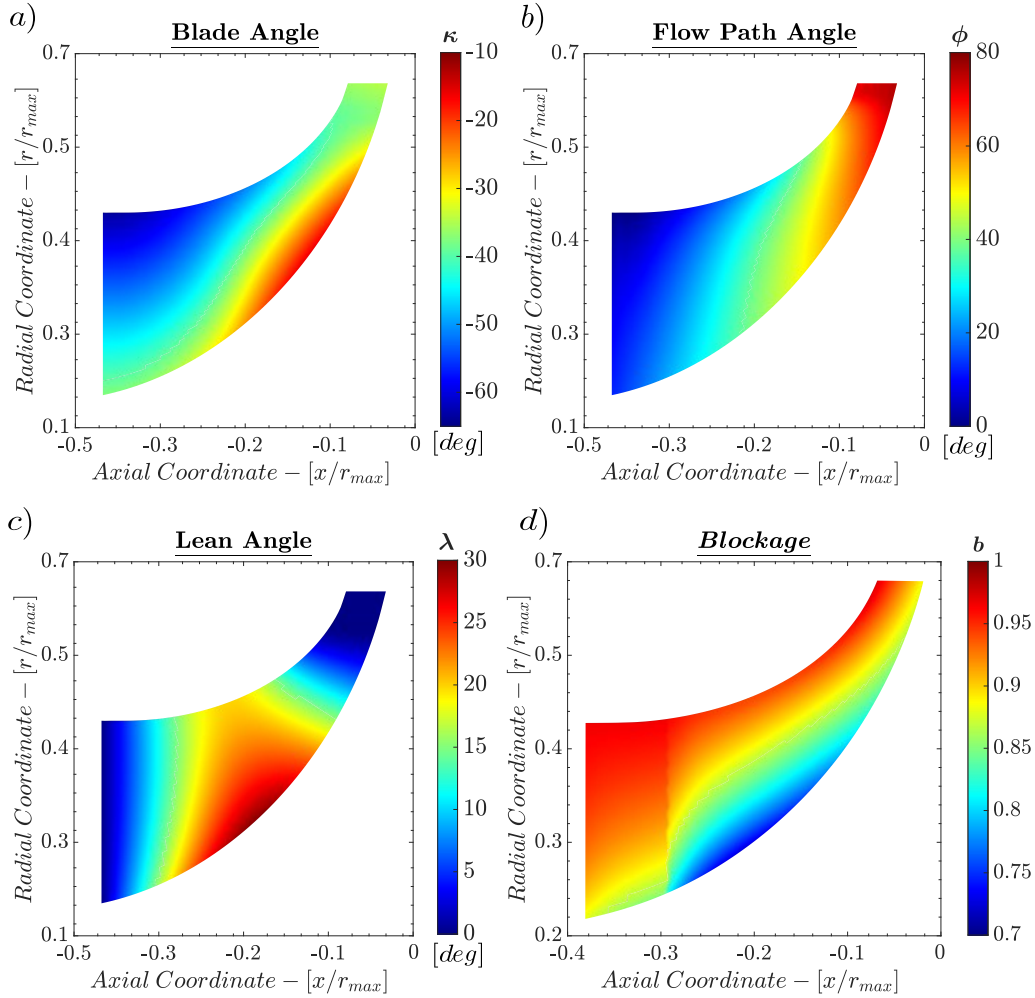


Figure 5.24: Blade angles and metal blockage parameter contours of SRV2 impeller.

The metal blockage along with the blade, flow path and lean angle distributions of impeller SRV2 blade are shown in figure(5.24). The magnitude of both the blade angle as well as their distribution pattern at the LE are similar to the axial rotor ones presented earlier, while the curvature of the camberline is uniform at the TE. The large variations between the inlet and the outlet blade metal angles indicate that energy addition in the flow is the combined outcome of both flow turning and changes in the mean streamline curvature. The flow path angle gradually increases in the streamwise direction, reaching values close to 80° at the TE of the impeller. This indicates a primarily radial flow at the outlet of the diffuser. The lean angle distribution reveals the presence of substantial lean towards the direction of rotation (positive angles) in the middle of the blade passage which diminishes significantly in the vicinity of the LE and the TE. A step-change increment in the blockage parameter due to the presence of the splitter blade can be observed in figure(5.24d). Compared to axial rotors, impeller blades are substantially thicker not only at the root but along

the entire spanwise direction, highlighting the significance of blockage modelling in centrifugal compressors.

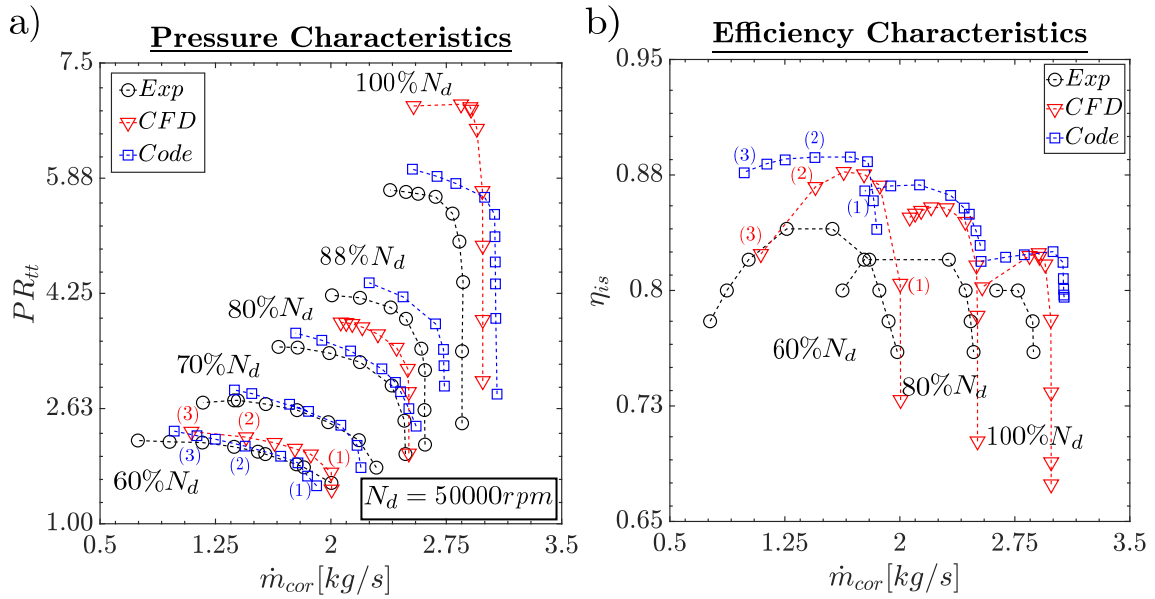


Figure 5.25: Comparison of pressure ratio and efficiency characteristics of SRV2-0 compressor with vaneless diffuser obtained by CFD and through-flow simulations.

The pressure ratio and efficiency characteristics predicted by the code and CFD are compared against the corresponding experimental characteristics and are depicted in figure(5.25). The characteristics correspond to different rotational speeds in the range [60% – 100%] of the design speed. Efficiency characteristics are plotted only for 60%, 80% and 100% design speed for readability purposes. The pressure ratio predictions obtained by the code are in excellent agreement with the experimental data in the lower [60% – 80%] range with the errors not exceeding 3.3%. The pressure ratio overestimations are more pronounced on the higher speed characteristics, where the peak pressure ratio is overpredicted by 5.6% and 6.4% at 88% and design speed respectively. The choking mass flow at design speed is also overpredicted by 7.65%. The overestimation of the choking mass flow is presumably the outcome of the high aerodynamic blockage which is significantly underpredicted by Aungier’s contraction ratio approach. According to Eisenlohr et al.[107], the estimated experimental aerodynamic blockage is around 17%, while the predicted one (using Aungier’s contraction ratio approach) is less than 5%. The higher pressure ratio predictions are predominantly related to the underestimation of the losses and increased flow turning, leading to higher efficiencies as shown in figure (5.25b).

The predictions of the CFD model exhibit a similar behaviour, with the pressure ratio being consistently overpredicted, while the mismatch is exacerbated in the higher

speed characteristics. The maximum pressure ratio errors of the CFD model are 5.8%, 10.9% and 19.3% for the 60%, 80% and 100% characteristics respectively, while the choking mass flow at design speed is overpredicted by approximately 7%. A mismatch of the same extent of both the mass flow and the pressure ratios at higher speeds is reported in several studies [107, 124] where similar results were obtained for the design speed pressure ratio characteristic using different 3D CFD codes. Eisenlohr et al. [107] concluded that the mismatch could be related to limitations of the turbulence models but no reasonable explanation for this phenomenon could be provided. The efficiencies are in good agreement with the experimental results, with the shape and the efficiency predictions of the CFD model being more accurate than the ones derived by the code. The errors do not exceed 4.4%, indicating that the high pressure ratio estimations are predominantly related to increased flow turning. Overestimation of the efficiency was also reported by [107], with the CFD model exhibiting a behaviour similar to what is reported in the literature. Similar results were obtained with a more refined mesh, thereby further investigation of the sources of errors was deemed redundant.

To assess the capability of the code to reproduce essential flow features, spanwise profiles of pressure ratio, temperature ratio and efficiencies at the TE are shown in figure (5.26). These correspond to the three annotated operating points along the 60% NRT characteristic (figure(5.25)) where the predictions of both models are more accurate. Due to flow separation at the shroud, the code was more prone to numerical instabilities when deviation distribution coefficients were applied, since they imposed excessive deviation angles at the shroud, which compromised the ability of the code to maintain locally control of the flow. This will be solved in the future by averaging or filtering out excessively high local deviation angles.

The pressure ratio profiles, extracted from CFD simulations at all three operating conditions, exhibit very similar patterns and are in good agreement with the profiles of the through-flow simulations. The increment in pressure ratio near the hub region along with the flatter profile in the largest part of the passage is captured reasonably well by the code. The large pressure ratio drop near the shroud, stems from the combined effect of the separated flow (reduced flow turning) and the presence of intense viscous phenomena, with the first being the most dominant one, as suggested by the high temperature ratio and low efficiency distributions in that region. The separation of the flow near the shroud is captured in the flow angle distributions, shown in figure(5.27). The trend and the shape of the temperature ratio profiles obtained by the code correlate well with the CFD ones, with the most notable

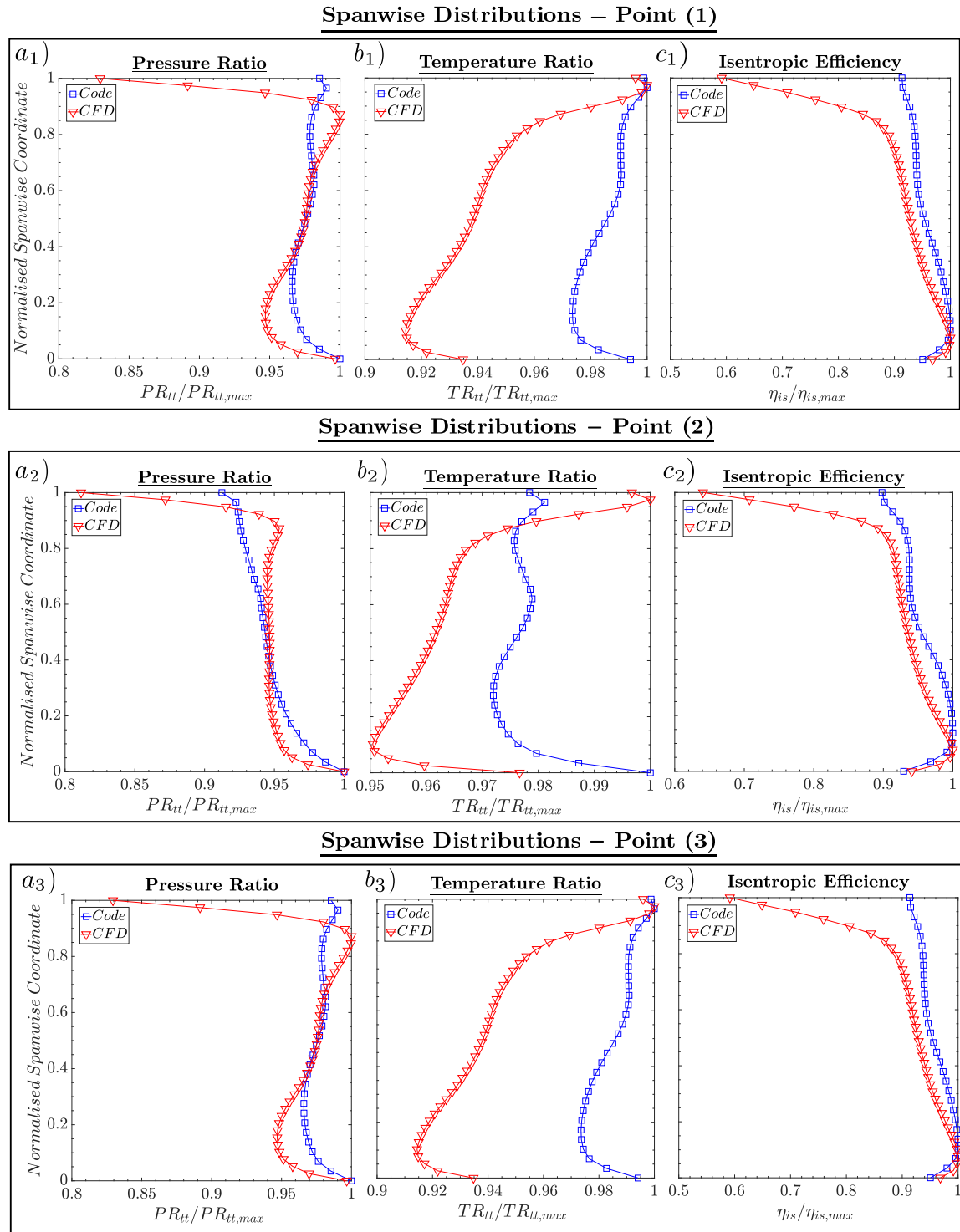


Figure 5.26: Comparison of total pressure ratio (top), total temperature (middle) and efficiency (bottom) profiles obtained by CFD and through-flow simulations of SRV2-0 compressor at three different operating conditions along the 60% design speed.

difference being the increase in the power input near the TE due to the high viscous forces which are underestimated by the through-flow code. The offset between the two curves is the outcome of the normalisation which in the case of the CFD curve,

is performed with a much larger temperature ratio. The presence of high entropy-generation phenomena near the shroud is evident by the efficiency drop, which can be observed in all efficiency profiles. The spanwise profiles of all quantities do not undergo any noteworthy changes as the operating point moves on the characteristic, implying that the inclusion of a single distribution coefficient for all operating points and characteristics would be a reasonably accurate modelling approach. The extension of this assumption to the deviation coefficient seems to be valid as suggested by the minor variations in the relative flow profiles shown in figure(5.27).

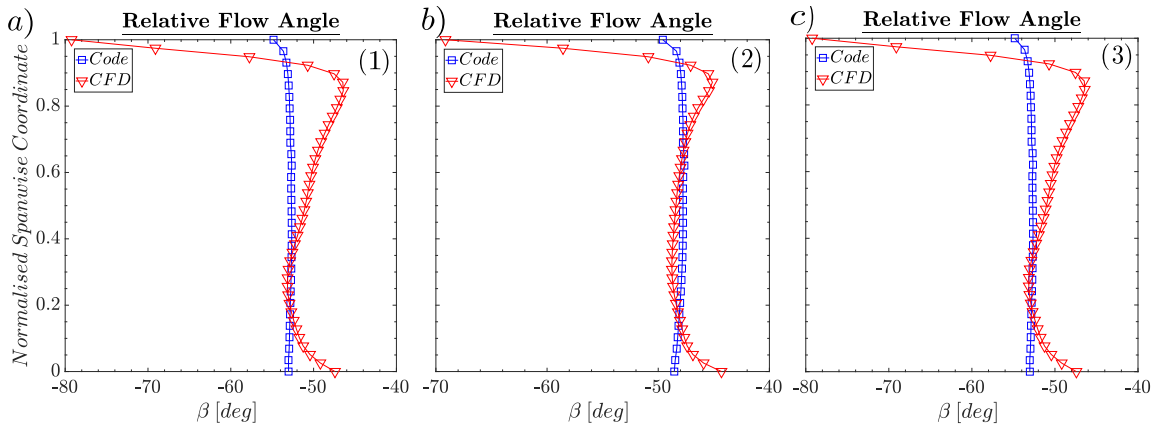


Figure 5.27: Spanwise distribution of relative flow angle at the TE of SRV2-0 impeller at different operating conditions at 60% design speed.

While the accuracy of the spanwise distribution of flow-field variables essentially defines the inlet boundary conditions of the downstream blade, the presence of a vaneless diffuser is expected to substantially modify the flow-field due to both diffusion of the flow as well as viscous phenomena e.g. mixing, boundary layers etc. These viscous phenomena cannot be naturally reproduced by an inviscid solver since the vaneless diffuser in the code is essentially modelled as an inviscid duct. The variations of the spanwise profiles presented earlier, at the TE, in the middle of the vaneless diffuser and the outlet are shown in figure(5.28).

The mixing-out of the separated flow at the shroud, combined with the thickening of the boundary layer at the hub and the shroud due to diffusion of the flow, result in a concave profile characterised by high loss regions near the hub and the shroud (figure(5.28 a_3, c_3)). In the diffuser, no energy exchange takes place (adiabatic walls) thus the temperature ratio profiles become more uniform due to mixing of the flow, especially near the hub and the shroud. Despite the aforementioned limitations, the profiles agree reasonably well with the CFD data, since the discrepancies are rather small and are magnified due to the small scale used in the x-axis. It is hypothesised,

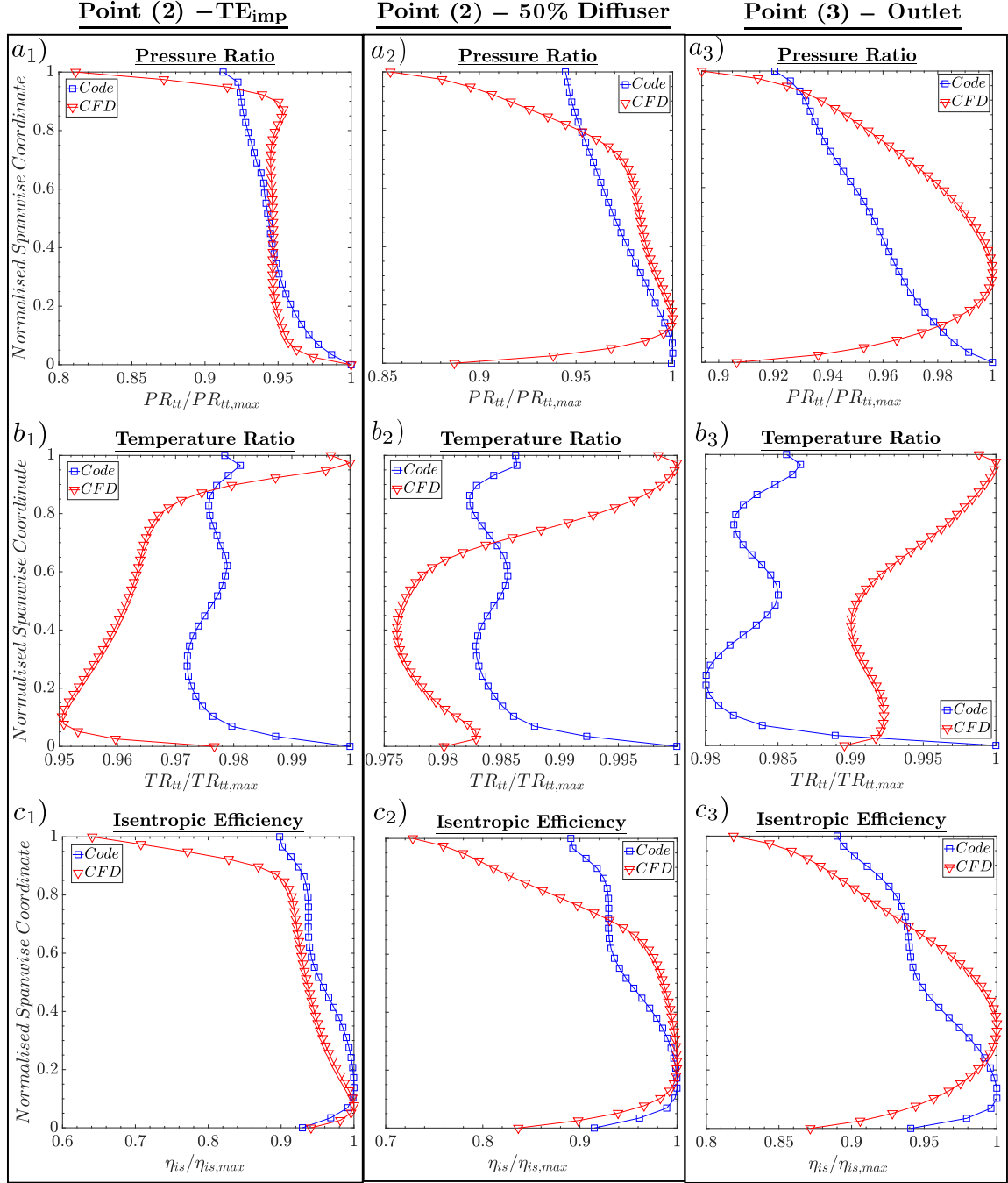


Figure 5.28: Efficiency, total pressure and temperature ratio spanwise distributions at the TE, middle of the vaneless diffuser and outlet of the SRV2-0 compressor at operating point (2) of the 60% design speed characteristic.

that the inclusion of losses in the diffuser, coupled with a viscous force coefficient and the inclusion of the artificial mixing model (not used in these simulations) could further ameliorate the aforementioned shortcomings.

To qualitatively assess the flow-field, contours of total pressure, temperature as well static pressure and meridional Mach numbers obtained by the code are

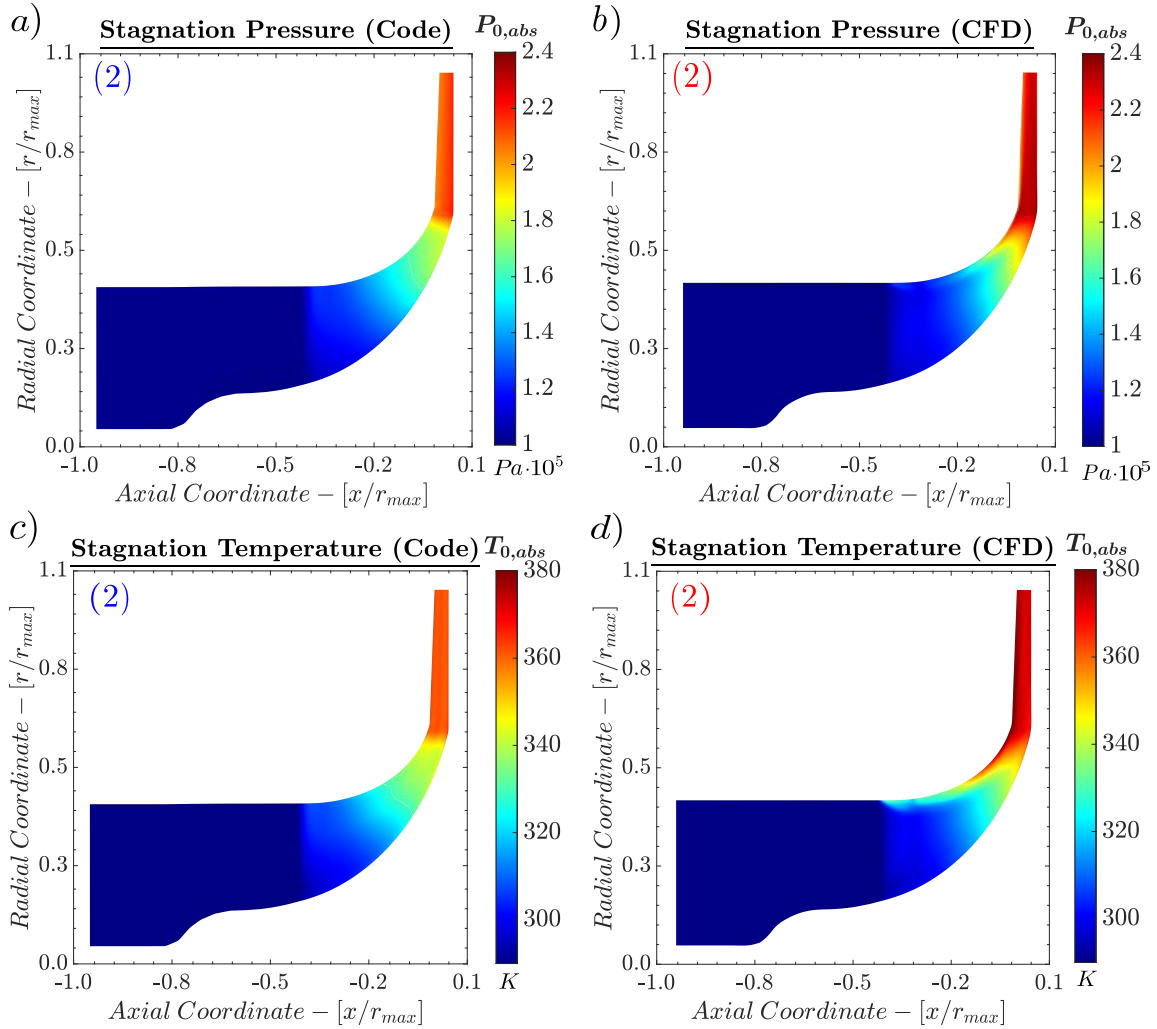


Figure 5.29: Comparison of total pressure and temperature contours of SRV2 impeller at operating point (2) of the 60% design speed characteristic.

compared against CFD results. The total pressure and temperature contours are in good agreement with the CFD results, successfully capturing the main flow-features (figure(5.29)). Some notable discrepancies are observed in the region near the shroud, due to presence of large viscous forces caused by the separated flow which are reflected on the increased temperature ratio near the shroud. The separation region initiates at the shroud, occupying a large fraction of the blade passage in the streamwise direction. This imparts the more non-uniform distribution of the total pressure and temperature CFD contours. The static pressure contours are in excellent agreement, while larger discrepancies can be seen in the meridional Mach number contours. In the vicinity of the LE, to avoid instantaneous flow turning and therefore instantaneous acceleration of the flow, the modified curvature of the blade (here set to 8%) gradually turns the flow causing an initial deceleration. This is evident by the lower meridional Mach

numbers near the LE (figure(5.30c))).

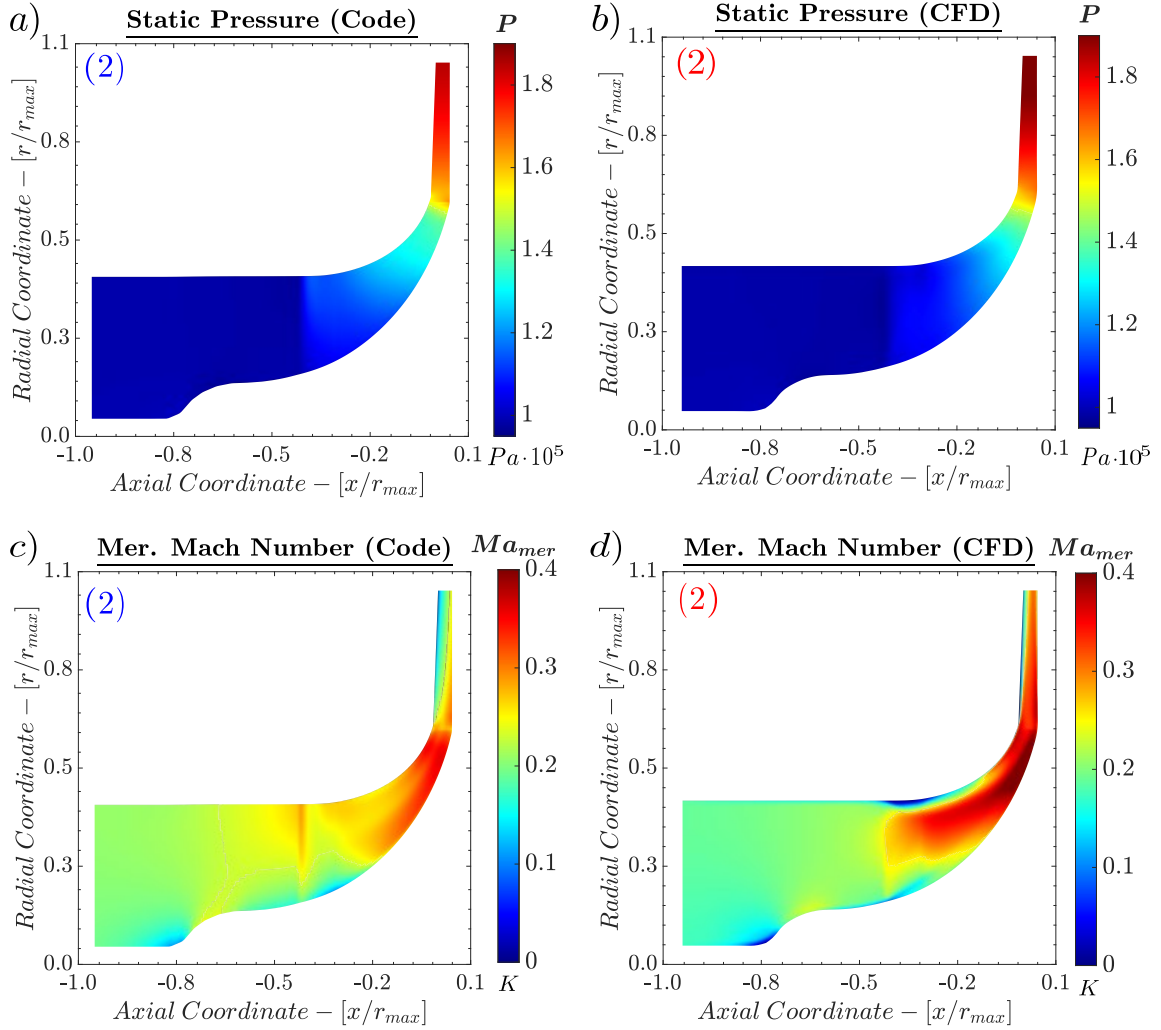


Figure 5.30: Comparison of static pressure and meridional Mach number contours of SRV2 impeller at operating point (2) of the 60% design speed characteristic.

The low Mach number region, at the shroud near the LE, indicates the separated flow region caused by the high incidence of the incoming flow at the TE. This is an inherently viscous phenomenon and as discussed previously, it cannot be naturally reproduced by the code, unless the imposed viscous forces are high enough to decelerate the flow and force it to separate or a high deviation angle is imposed via the distribution coefficient. The separated flow region near the shroud is accompanied by an enhanced viscous dissipation, which in turn requires larger energy input by the blade to push the flow (figure(5.30d)). In the diffuser, higher Mach numbers are predicted by the CFD model with the high Mach number region being substantially more extended and uniform in the spanwise direction in the CFD contour due to flow mixing. The low Mach number region near the shroud in the vaneless diffuser,

indicates flow separation due to insufficient shroud curvature smoothness of the shroud geometry near the TE of the impeller. This is a deficiency of the compressor design and not of the modelling/grid generation approach.

5.1.4 NASA CC3 - Forward flow

To evaluate the ability of the code to model more complex centrifugal compressor geometries incorporating vaned diffusers and volutes, the NASA CC3 compressor was modelled initially with a vaneless and subsequently with a vaned diffuser [119]. A 3D visualization of the two compressor configurations is shown in figure(5.31). The NASA CC3 compressor is essentially an up-scaled Alison compressor design with a design mass-flow almost 3 times higher than the original design (4.54 kg/s instead of 1.658 kg/s) and is the largest centrifugal compressor tested within the present work. Two CFD models are developed with the the first accommodating a vaneless diffuser, while the latter a wedge-type vaned diffuser. Both configurations are modelled in the code and the results of both models are compared against experimental pressure ratio and efficiency characteristics. Following a similar approach, the precision of the code's flow-field predictions are evaluated both quantitatively as well as qualitatively by comparing spanwise profiles and flow-field contours of the quantities presented in the previous section at different operating points and positions.

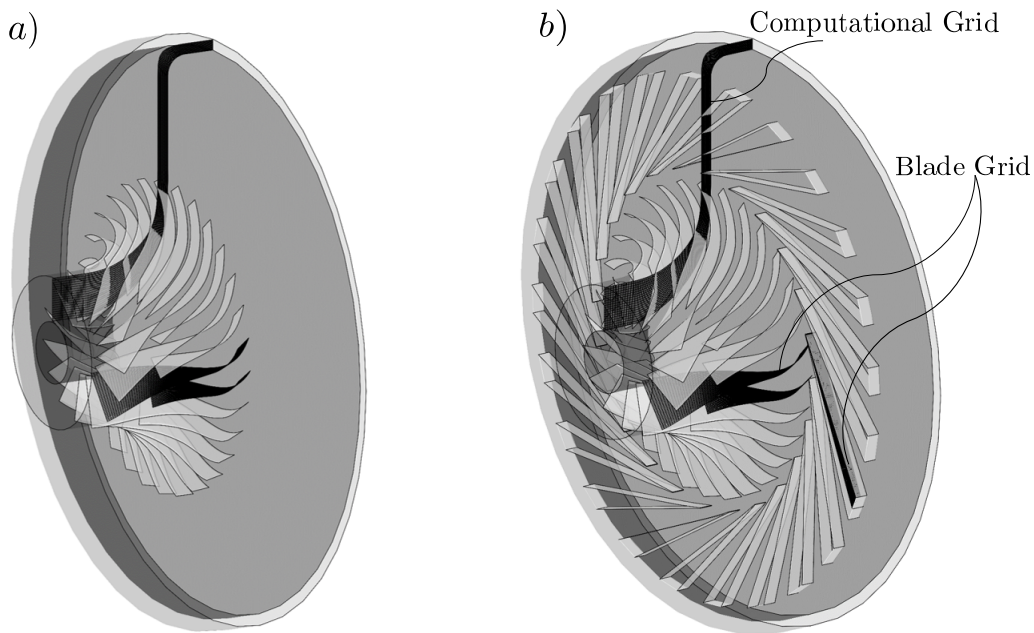


Figure 5.31: 3D visualization of centrifugal compressor NASA CC3 a) with vaneless and b) without vaned diffuser. The main computational duct grid along with the camber surface grid is also shown.

The steady-state, CFD solution of the flow-field is obtained by performing 3D, single-passages simulations. The vaneless configuration consists of a single domain, encompassing a full and a splitter blade along with the circumferential sector corresponding to the inlet, the vaneless diffuser and the volute. Regarding the second compressor configuration, a single passage mesh of the vaned diffuser comprising a single diffuser vane, a part of the vaneless space and the volute, is numerically adjoined to the impeller blade passage via a mixing plane (figure(5.32)). Naturally, in that configuration the impeller blade passage domain extends up to the vaneless space. The first configuration comprises approximately 10 million elements, while the latter approximately 12 million elements. The grid was generated in ANSYS Turbogrid, while the $\kappa - \omega SST$ turbulence model was preferred, following the guidelines in [125]. The solution of the flow-field is performed in CFX using second order-accurate schemes for the advection and turbulence terms. The inclusion of a mixing plane is necessary only in the second configuration with the vaned diffuser.

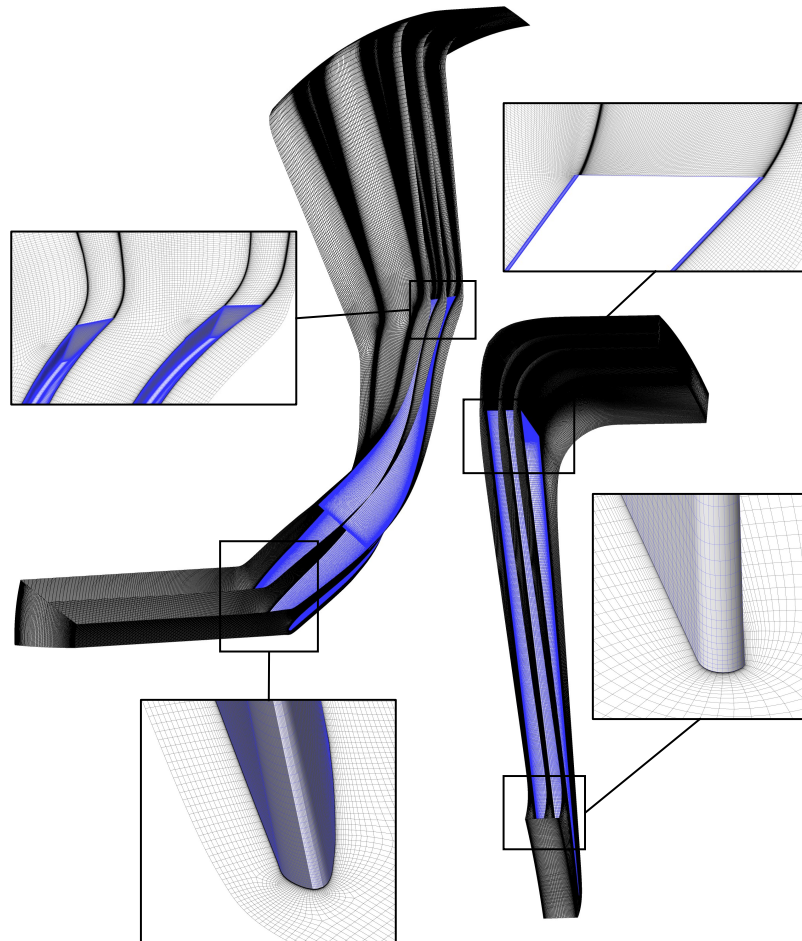


Figure 5.32: Overview of NASA CC3 impeller and vaned diffuser computational domains used for single-passage RANS simulations.

Figures(5.33) and (5.34) depict the discretized pressure, suction and camber surfaces of the full impeller and the vaned diffuser blade as well as the meridional projection of the camber surface grids. Similar to impeller blades, it is necessary to discretise vaned diffuser blades with a larger number of elements in the streamwise direction to ensure high enough grid resolution due to the high aspect ratio of the blades. It should be noted that the streamwise direction of the vaned diffuser blade essentially coincides with the radial axis. Both blades are discretised with 70 streamwise and 50 spanwise elements, consisting of 3500 elements.

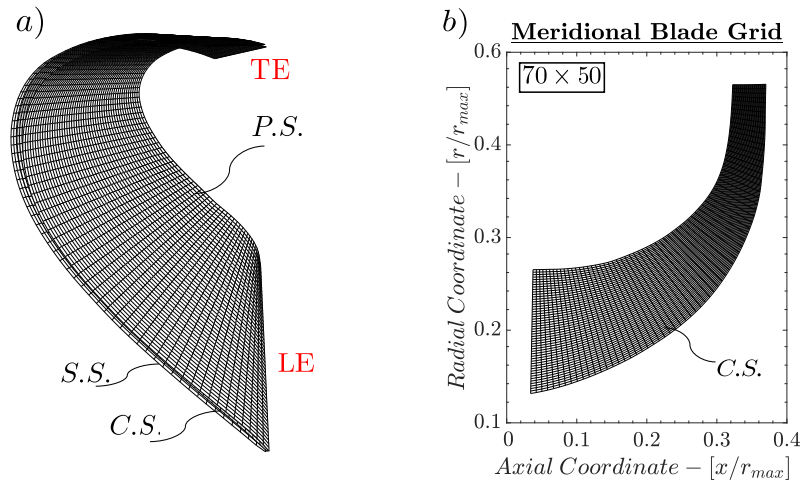


Figure 5.33: Overview of NASA CC3 impeller a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.

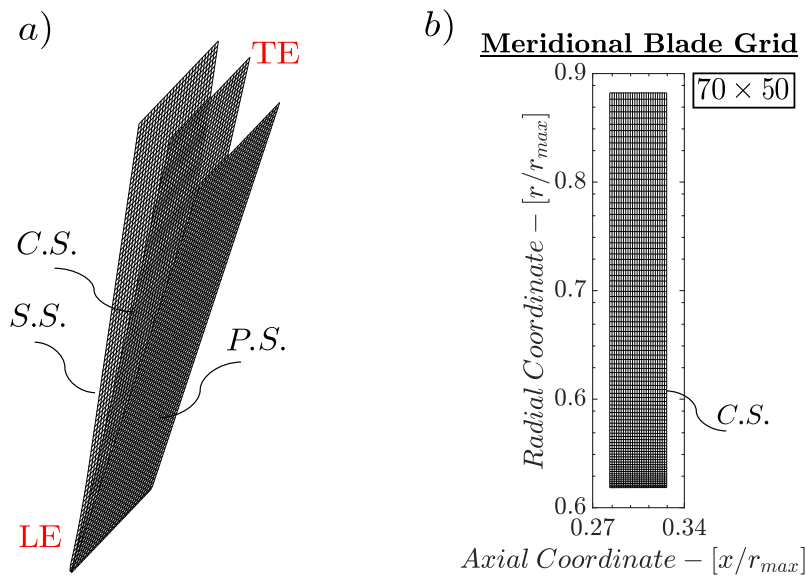


Figure 5.34: Overview of NASA CC3 vaned diffuser a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.

The flow-field solution of the code is derived on the computational grid shown in figure(5.35). The grid consists of 5 subdomains; the inlet, the impeller, the vaneless space, the vaned diffuser blade and the volute. The computational grid is generated using the adaptive mesh approach with a total element count of 7740. The impeller and the vaned diffuser are discretised using 160 and 96 elements in the streamwise direction, while less elements are used in the spanwise direction due to the narrower size of the duct (20 compared to 30 for the SRV2). The large changes in radius at the vaneless space, diffuser and volute subdomains, induce substantial increments in the cell face areas and volumes, thus the dense mesh resolution is maintained even downstream the TE of the vaned diffuser despite the absence of body-force fields. The initial compressor configuration with the vaneless diffuser is solved on the same computational grid by deactivating the blockage and the body-forces in the vaned diffuser domain. Uniform distribution coefficients are used for both the viscous force and the deviation.

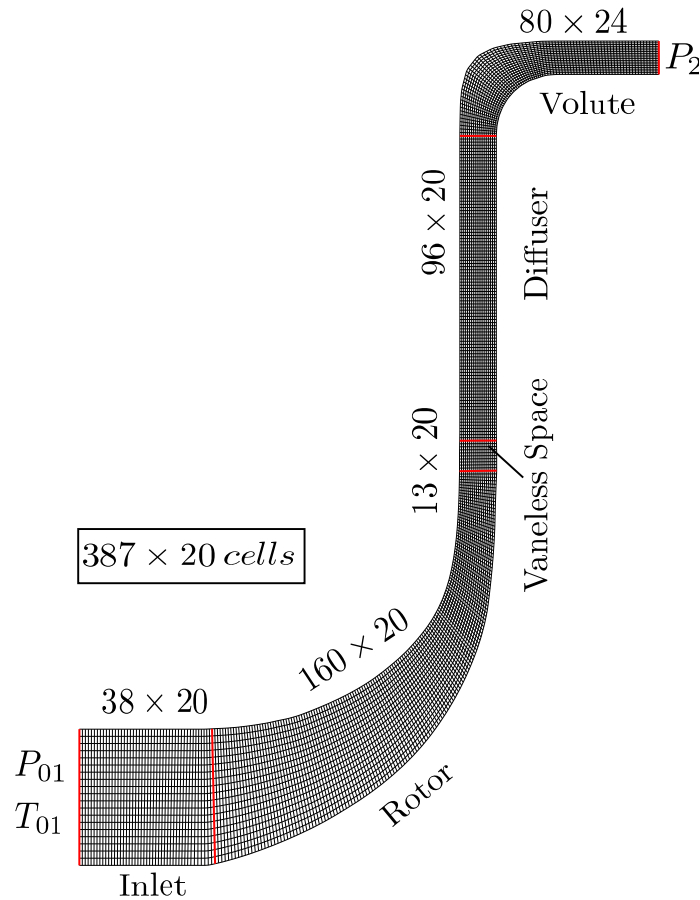


Figure 5.35: Computational domain of NASA CC3 compressor generated by the code using the adaptive mesh approach.

Figures(5.36) and (5.37) show the blockage and blade angle distributions of the NASA's CC3 impeller and vaned diffuser blades. The distribution of the blade angle exhibits similar characteristics to the SRV2; the blade angle gradually increases towards the shroud at the LE and small blade angles can be seen near the hub in the middle of the blade. The blade angles at the TE are uniformly distributed and are comparable to the LE ones, indicating that the power input is dependent primarily on the change in radius and not on flow turning. As anticipated, there is a close resemblance of NASA's CC3 flow-path angle distributions to the SRV2 although slightly higher angles are recorded near the TE, reaching approximately 90° at the TE. Contrary to impeller SRV2, substantial blade leaning towards the direction of rotation can be observed in the vicinity of the TE, increasing locally the circumferential turning force components and consequently the energy input. Substantial reduction in the effective area by up to 35% can be seen along the entire passage, further highlighting the significance of metal blockage modelling in centrifugal compressors.

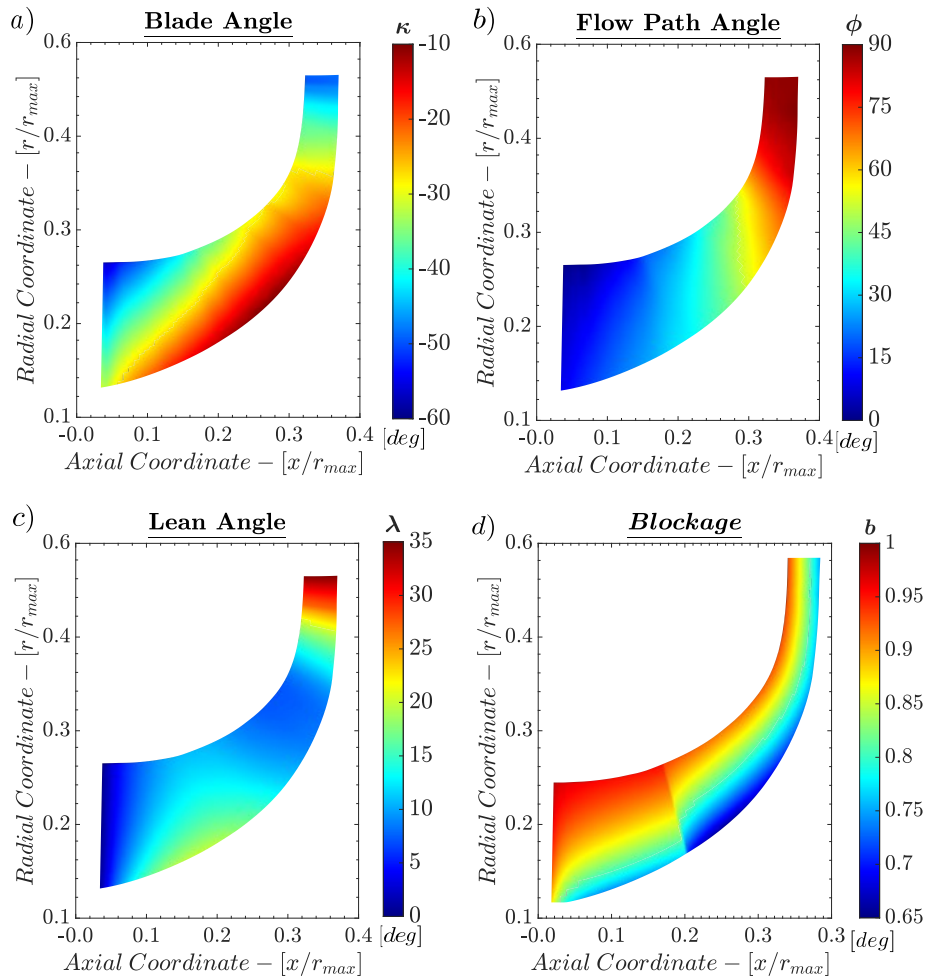


Figure 5.36: Blade angles and metal blockage parameter contours of NASA CC3 impeller blade.

Due to the simplified, wedge-type design of the vaned diffuser blades, the spanwise distribution of the blockage and the angles is uniform. The lean angle is minuscule, while the flow-path angle is 90° along the entire vaned diffuser passage due to the radial position of the blade. The diffuser blades turn the incoming flow by almost 50° . The large thickness of the vanes introduces excessive metal blockage reducing the effective area by up to 40% near the TE. To prevent numerically-induced errors due to the sudden change in the size of the cell face areas and volumes once the flow exits the vaned diffuser subdomain, the blockage at the TE is modified via custom cubic shape functions as described in section(4.1.3).

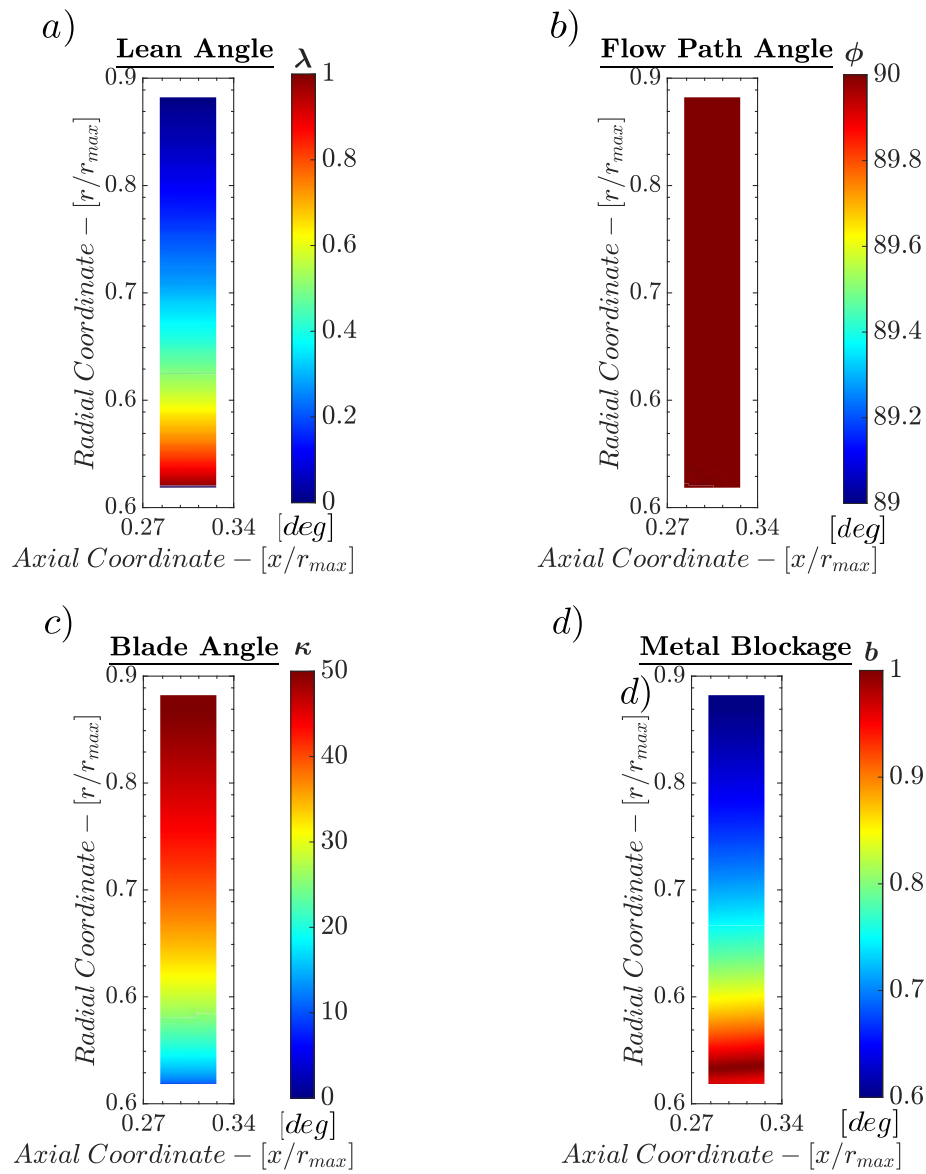


Figure 5.37: Blade angles and metal blockage parameter contours of NASA CC3 vaned diffuser blade.

Aerodynamic performance predictions for both compressor configurations are derived with both the code and the CFD model and are compared against experimental data in figures(5.38) and (5.39). The pressure ratio characteristics obtained with the CFD model are in excellent agreement with the experimental results. For clarity only the 60%, 80% and 100% efficiency characteristics are plotted. The characteristics obtained with the through-flow code are in very good agreement with the experimental data along the entire speed range. The pressure ratio is consistently underestimated in the higher mass flow region but correlates exceptionally well in the lower one. The lower pressure ratio predictions are associated with the overestimation of the pressure losses by the empirical loss model which is reflected on the lower efficiency predictions in the higher mass flow region as seen in the 60% and 80% efficiency characteristics in figure(5.38b). The maximum errors in the higher mass flow regions reach approximately 9% but reduce to less than 2% in the higher mass flow regions. At design speed, the code slightly overpredicts the efficiencies which is reflected on the alleviation of the pressure ratio errors observed in the lower speed range. The efficiency errors of both models are comparable, however the shape of the characteristic is better approximated by the CFD model.

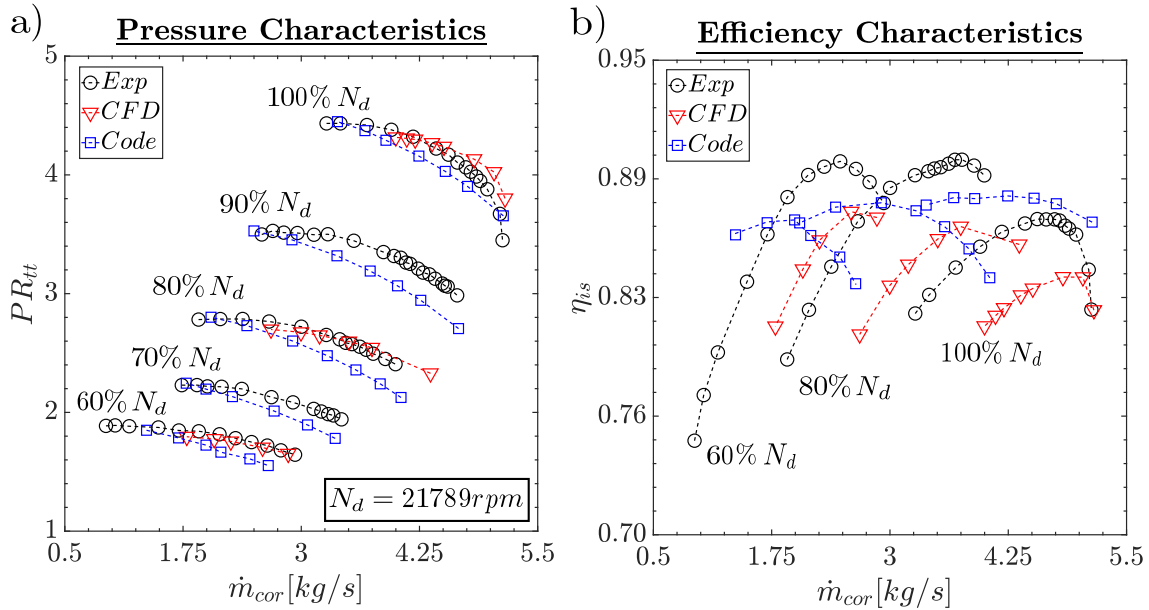


Figure 5.38: Comparison of pressure ratio and efficiency characteristics of NASA CC3 compressor with vaneless diffuser obtained by CFD and through-flow simulations.

Figure(5.39) shows the 60%, 80% and 100% speed pressure and efficiency characteristics corresponding to the second configuration of the NASA CC3 compressor with a vaned diffuser. Characteristics derived from through-flow and CFD simulations are compared against experimental data. The CFD pressure characteristics are in

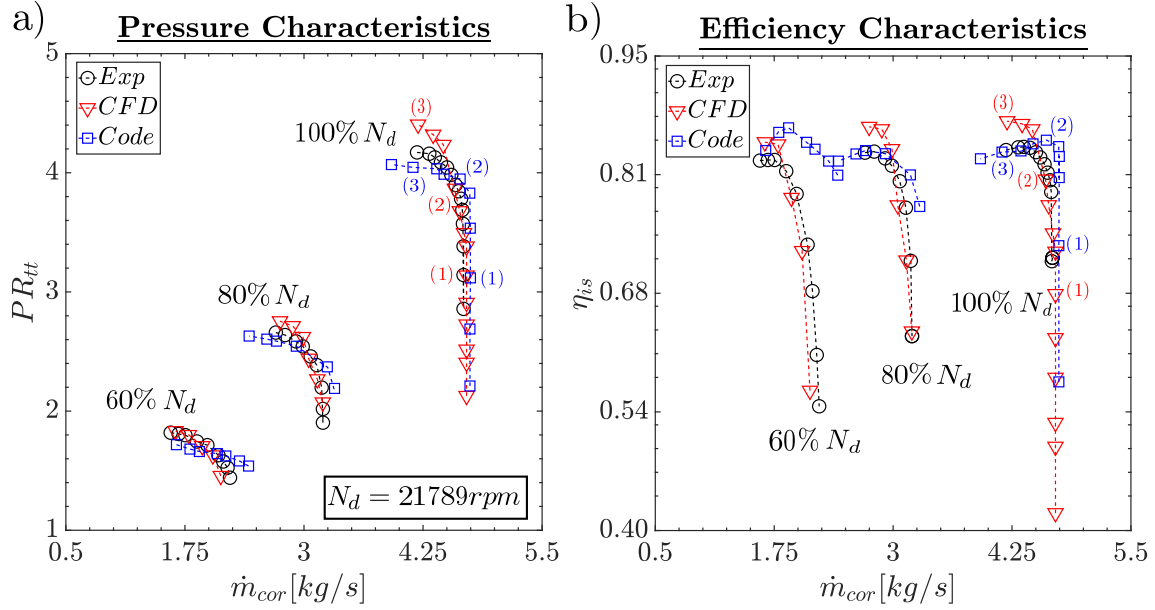


Figure 5.39: Comparison of pressure ratio and efficiency characteristics of NASA CC3 compressor with vaned diffuser obtained by CFD and through-flow simulations.

excellent agreement with the experimental results. The peak pressure ratio is slightly overpredicted in the lower mass flow region by up to 4.4% and 5.5% at 80% and 100% speed respectively, while the error is less than 1% at 60% speed. The pressure ratio and the choking mass flow predictions obtained by the code at design speed, correlate exceptionally well with the experimental data. The maximum pressure rise is underpredicted by 3.7%. The predictions are less accurate at 60% speed, where the maximum pressure rise is overpredicted and the shape of the characteristic is less consistent with the experimental one. This indicates erroneous estimation of the mass flow capacity potentially due to aerodynamic blockage effects, misestimations of the slip factor, since the efficiencies are in agreement with the experimental data, as well as upstream influence induced by the blades which cannot be replicated by body-force methods (see figure(2.18)). Inaccuracies in the higher mass flow regions were also observed in the studies of Kottapalli and Benneke [49, 85]. It should be noted that all body-force simulations are performed with a 9% blockage, as predicted by Aungier's correlation. The efficiency estimations of both the code and the CFD model are in very good agreement with the experimental data. The CFD model slightly overestimates the peak efficiencies at 80% and 100% speed by approximately 3.2%, while the corresponding errors of the through-flow code characteristics are less than 1.2%.

Since the viscous force and the deviation angle are distributed in the blade passage using standard, uniform distribution coefficients, spanwise profiles of global aerody-

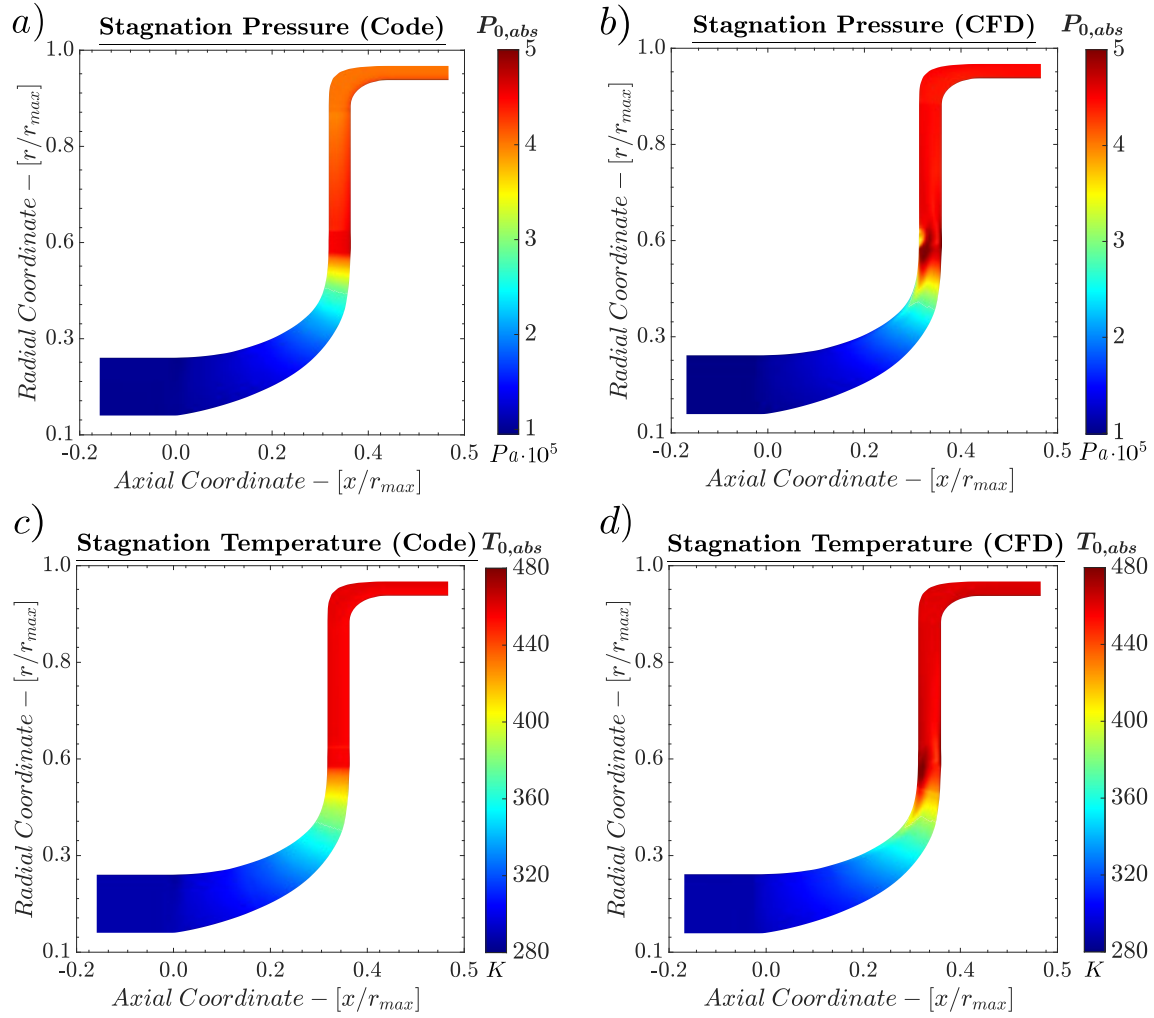


Figure 5.40: Comparison of NASA CC3 compressor (vaned diffuser) stagnation pressure and temperature contours obtained by the through-flow code and the CFD model.

dynamic performance quantities are not presented. The assessment of the flow-field is performed qualitatively by comparing contours of the flow-field variables presented in the previous subsections. Figure(5.40) shows total pressure and temperature contours computed by the code and the CFD model corresponding to operating point (3) on the 100% characteristic of the configuration with the vaned diffuser. The contours obtained with the code agree exceptionally well with the CFD ones while some minor discrepancies can only be observed in the area upstream and downstream the TE (vaneless space).

Figure(5.41) shows a comparison between static pressure and meridional Mach number contours. The static pressure contours correlate remarkably well, with the most notable difference being the slightly higher pressure values of the CFD model. This is consistent with the higher total pressure rise, reflected on the pressure characteristics in figure(5.39). The meridional Mach number contours are in very

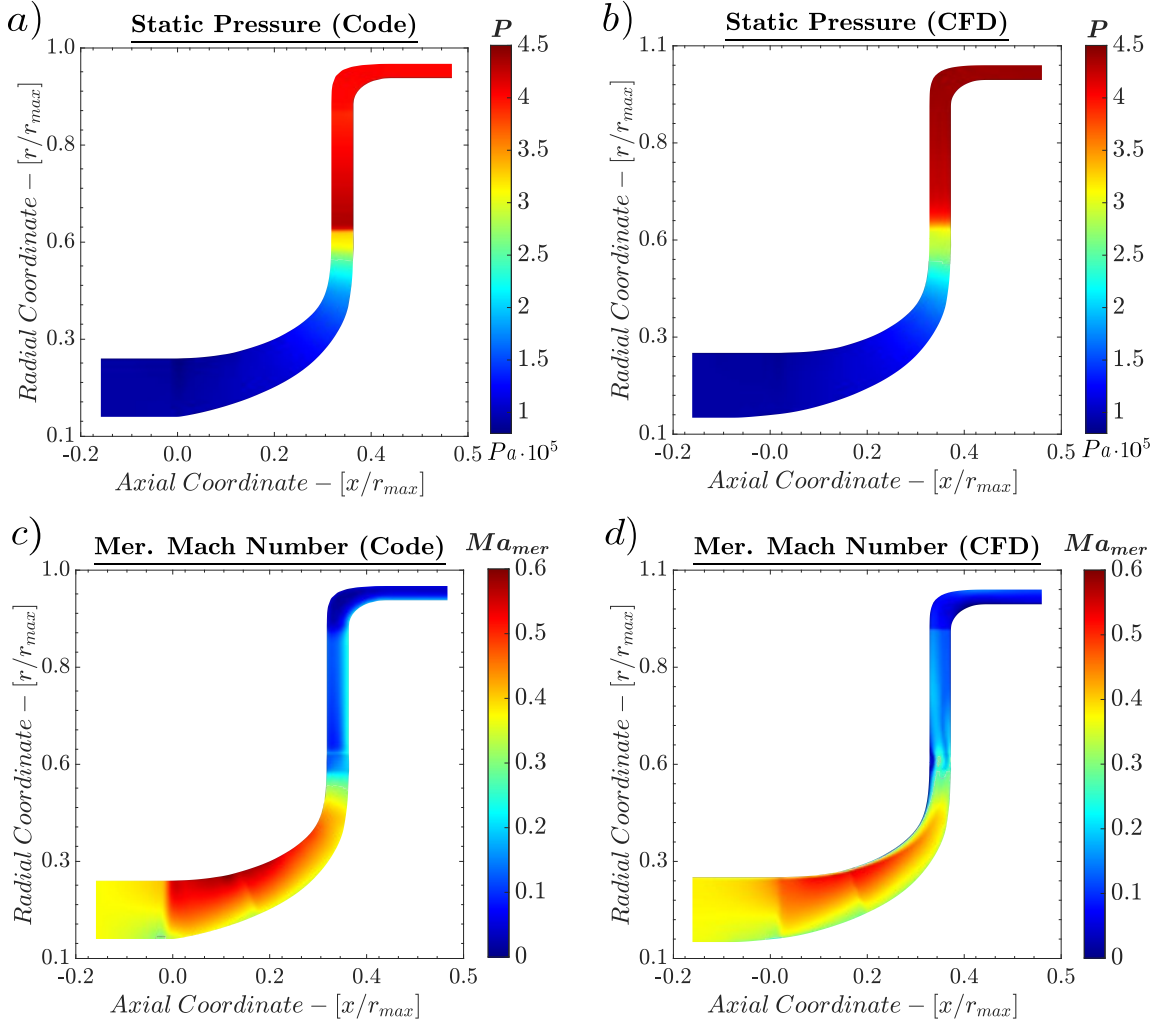


Figure 5.41: Comparison of NASA CC3 compressor (vaned diffuser) static pressure and meridional Mach number contours obtained by the through-flow code and the CFD model.

good agreement with the CFD ones. The through-flow code successfully reproduces the meridional Mach number distribution at the splitter blade LE caused by the sudden increase in the blockage (via the blockage parameter). The lower momentum region near the impeller hub due to boundary layer growth cannot be accurately captured by the through-flow code since a uniform distribution coefficient is used. Downstream the LE of the splitter blade and near the shroud, a small separation region can be observed which cannot be captured by the code due to the imposition of uniform distribution coefficients and due to the inherently viscous-related nature of the phenomenon. This mixes-out in the vaneless diffuser and the flow becomes uniform again once it enters the vaned diffuser blade. Except for that phenomenon, the Mach number magnitudes and distributions are accurately replicated by the code.

5.1.5 CSTAR - Forward flow

The ability of the code to model complex centrifugal compressor geometries is concluded with a final test case representative of a contemporary, lower mass-flow centrifugal compressor with thin, aerofoil-type vaned diffuser blades and a volute. A 3D visualization of the CSTAR compressor is shown in figure(5.42).

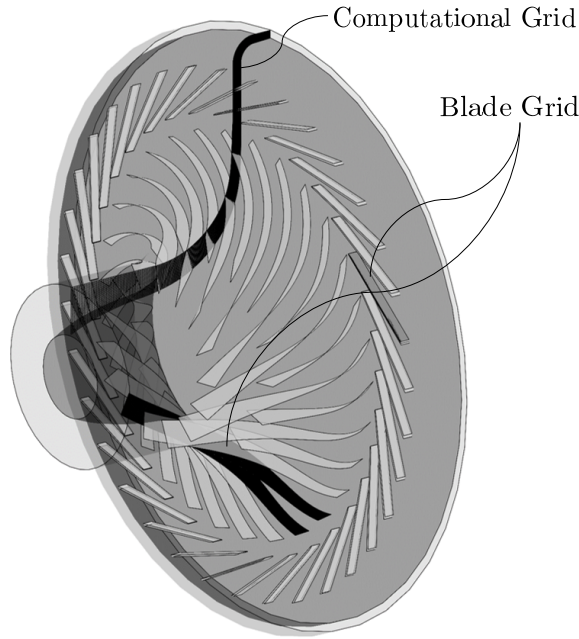


Figure 5.42: 3D visualization of CSTAR centrifugal compressor. The main computational duct grid along with the camber surface grid is also shown.

A CFD model, validated against experimental results, of the CSTAR compressor constitutes the benchmark hereafter to assess the precision of through-flow code predictions. Global aerodynamic performance predictions, spanwise profiles and flow-field contours of the quantities presented earlier at different operating points and positions are compared to conclude the forward-flow validation process of the developed body force-based methodology. Single passage, RANS simulations are performed using ANSYS CFX to obtain the reference data against which, the predictions of the code will be compared. The CFD computational domain consists of two domains adjoined via a mixing plane interface. The first encompasses the impeller passage consisting of a full and a splitter blade, along with the corresponding circumferential sector of the inlet and part of the vaneless space. The second computational domain comprises part of the vaneless space, a single vaned diffuser blade passage as well as the corresponding circumferential sector of the volute. Both computational domains are shown in figure(5.43). The entire grid consists of approximately 7 million elements. All simulations are performed using identical solver settings; $\kappa - \omega$ SST turbulence

model and second order-accurate schemes for the advection and turbulence terms.

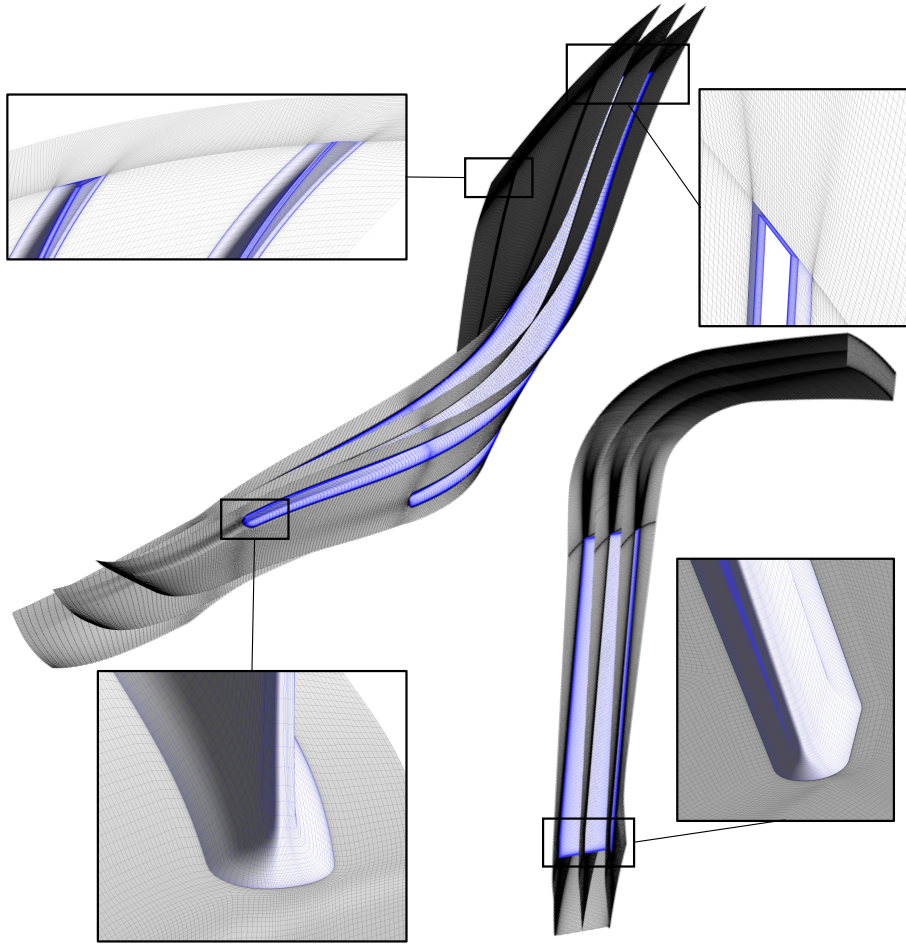


Figure 5.43: Overview of CSTAR impeller and vaned diffuser computational domains used for single-pass RANS simulations.

Figures(5.44) and (5.45) illustrate the discretised surface meshes of the impeller full and splitter blade as well the projection of the camber surface mesh on the meridional plane. Both blades are discretised with the same number of elements in both the streamwise and the spanwise direction as the one used for SRV2-0 and NASA CC3 impellers i.e. 70×50 . The definition of the curvature and the blockage parameter is performed following the approach described in the previous test cases, however contour plots of their distributions are not included since they constitute commercially sensitive data. Compared to NASA's CC3 vaned diffuser blades, the vaned diffuser blade of the CSTAR compressor follow a more sophisticated, aerofoil-type design, with increased camberline curvature and substantially reduced thickness.

The computational grid used for the through-flow simulations is shown in figure(5.46). The adaptive mesh approach is used, while a dense grid resolution is preserved downstream the impeller in non-bladed regions to prevent large changes in

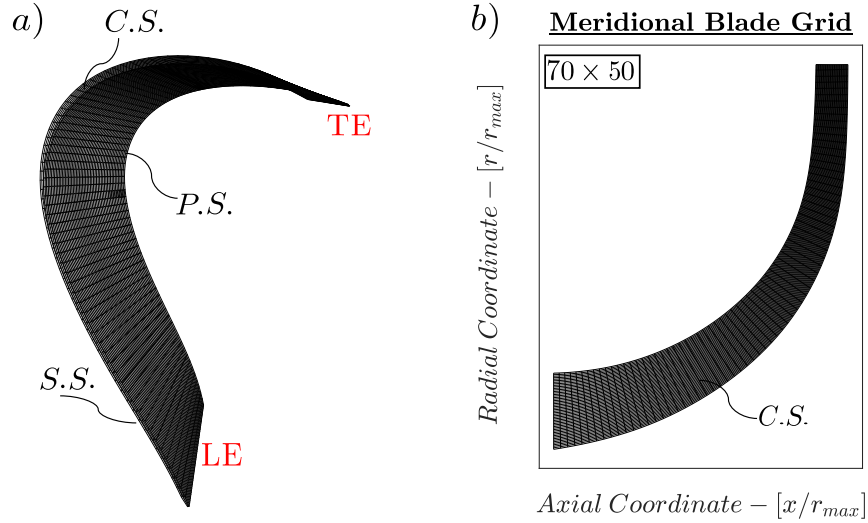


Figure 5.44: Overview of CSTAR impeller a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.

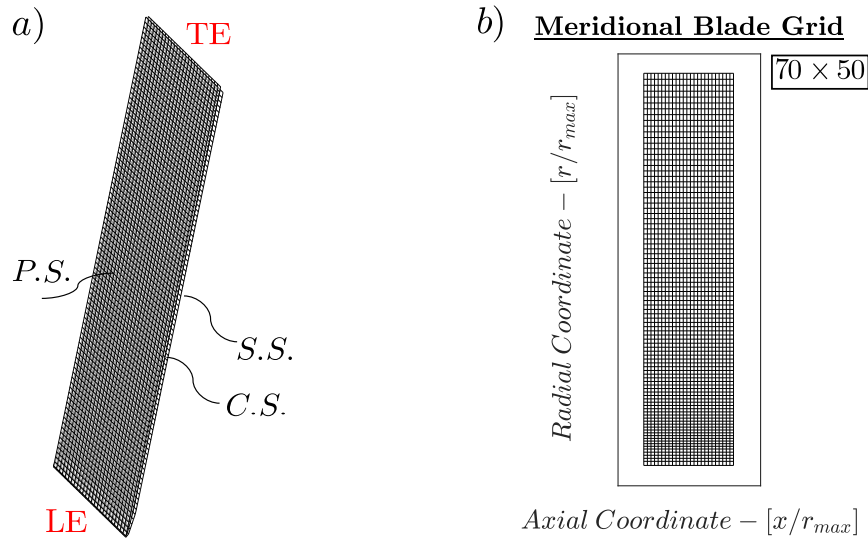


Figure 5.45: Overview of CSTAR vaned diffuser a) 3D blade grid consisting of the pressure, suction and camber surfaces and b) meridional projection of the camber surface mesh.

cell face face areas and volumes. Transition regions are added at the LE and TE of the vaned diffuser to match the lower aspect ratio cells in the vaned diffuser. The grid comprises 10380 elements, with the impeller and the vaned diffuser comprising 130 and 70 streamwise elements respectively.

A comparison between the pressure ratio and efficiency characteristics of the CSTAR compressor predicted by the CFD model and the through-flow code are presented in figure(5.47). The pressure ratio characteristics in the higher 80% and 100% speed range are in excellent agreement with the CFD results. The code underestimates the pressure ratio with respect to the CFD model by up to 2.8%,

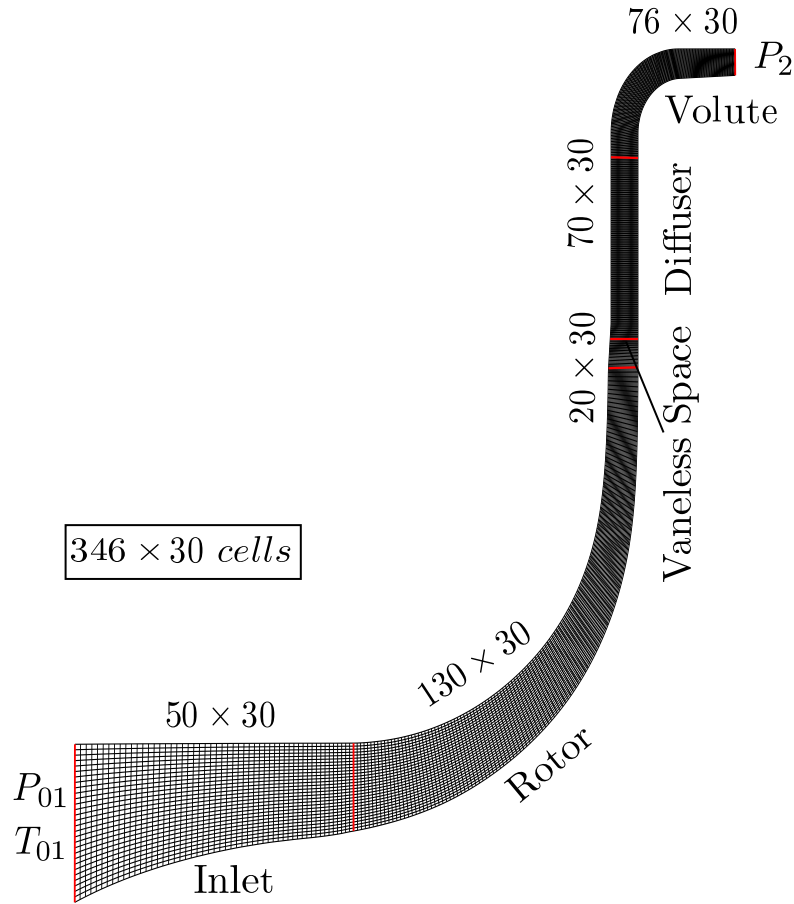


Figure 5.46: Computational domain of CSTAR compressor generated by the code using the adaptive mesh approach.

3.2% and 2.9% at 60%, 80% and 100% speed respectively. Higher discrepancies are observed in the higher mass flow region of the 60% characteristic regarding both the pressure ratio and mass flow as well as the inconsistent shape of the characteristic. It is hypothesized that this linked to the reasons mentioned in the analysis of the NASA CC3 configuration with the vaned diffuser. The efficiency characteristics correlate reasonably well with the efficiency predictions of the CFD model. The code consistently overestimates the efficiencies by up to 4.2% at 60% speed, while the maximum errors are approximately 7.5% at the higher speed characteristics. The results suggest a concurrent overestimation of flow slippage along with more modest pressure loss predictions; the implications on the pressure rise of the first are balanced out by the latter, however the underestimation of the pressure loss is perceivable by the higher efficiencies. It should be noted, that all results are obtained without any prior tuning of the loss and slip models.

Spanwise pressure ratio, temperature ratio and efficiency profiles at the TE of the impeller corresponding to the annotated points along the 80% characteristic are shown

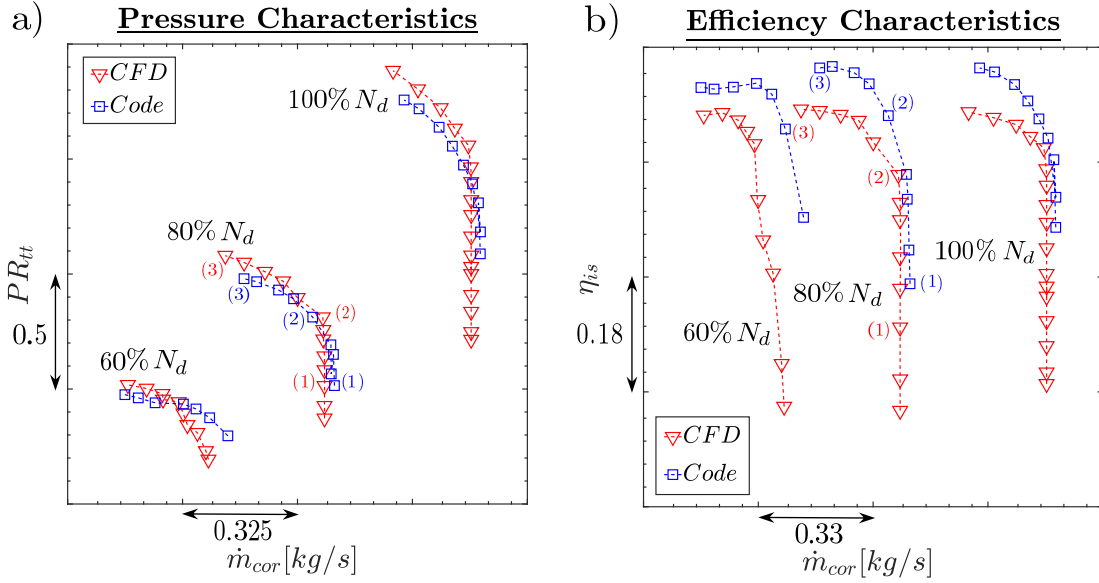


Figure 5.47: Comparison of pressure ratio and efficiency characteristics of CSTAR compressor obtained by CFD and through-flow simulations.

in figure(5.48). These distributions are similar to the ones presented in section(4.4) therefore the reader is referred to that section for a more in-depth analysis.

The pressure ratio profile is predicted reasonably well but is unable to accurately reproduce the reduced pressure rise near the hub and the shroud, a deficiency of the distribution coefficient approach which was also encountered in the previous compressor geometries. The consistency of the temperature ratio profiles near the shroud along with the higher efficiencies indicates that the source of the error is the combined outcome of reduced flow slippage (increases temperature ratio but not efficiency) and lower losses (increases temperature ratio and lowers efficiency). Near the hub, the mismatch is predominantly associated with the intensity of viscous phenomena, the extent of which is underestimated by the code. The trends of all quantities are nevertheless reproduced reasonably well at all three operating conditions, further justifying the idea of using a single distribution coefficient for the entire characteristic in impeller blades. The absolute flow angles downstream the TE, in the vaneless space are shown in figure(5.49). The flow angles are in very good agreement with the CFD results at all operating points with the largest discrepancies being confined with then hub and the shroud region, where the viscous effects are more pronounced.

The effect of flow mixing downstream the TE and its influence on the redistribution of the flow-field variables is investigated since it was hypothesized that flow mixing inside a non-diffusing duct would result in a relatively uniform distribution profile. In

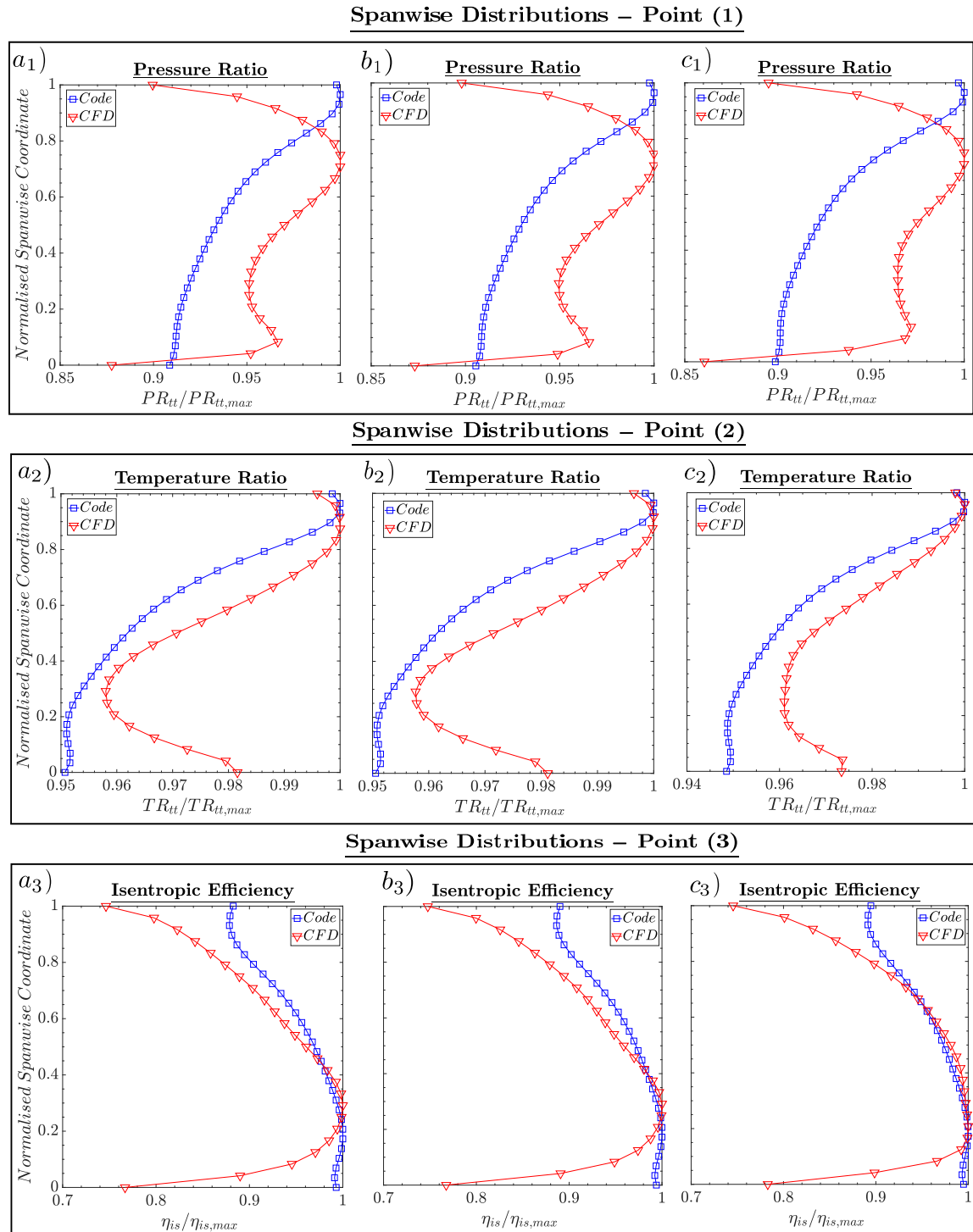


Figure 5.48: Comparison of total pressure ratio (top), total temperature (middle) and efficiency (bottom) profiles obtained by CFD and through-flow simulations of CSTAR compressor at three different operating conditions and 80% design speed.

an inviscid through-flow code, this could be resembled by imposing a simple uniform distribution coefficient to the blade located upstream the volute. Figure(5.50) shows the changes in the temperature ratio profile at the TE of the vaned diffuser, in the

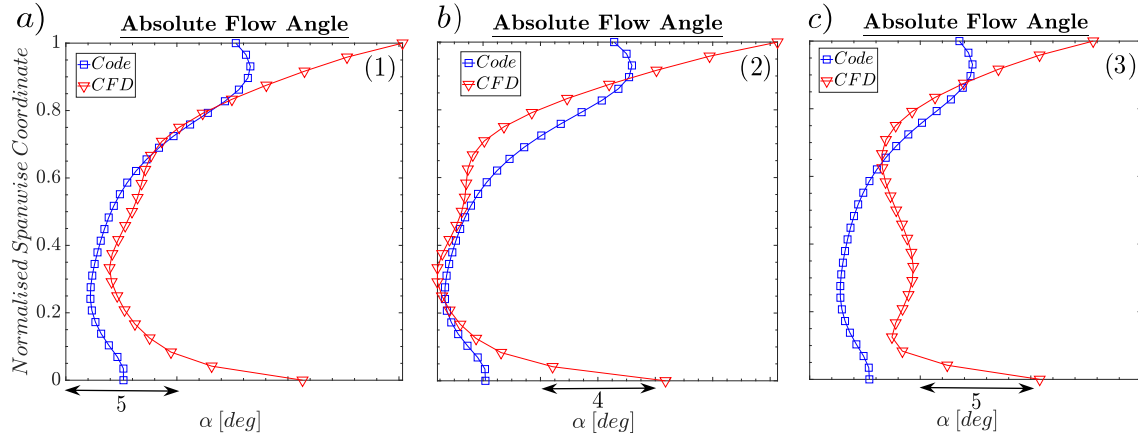


Figure 5.49: Comparison of absolute flow angle spanwise distribution in the vaneless space of CSTAR compressor.

middle of the volute as well as the outlet, obtained by the CFD model as well as the code, using uniform and CFD-derived distribution coefficients. The temperature ratio profile at the TE (figure(5.50a)) extracted from through-flow simulations coupled with the CFD-derived viscous force coefficient is in excellent agreement with the CFD results, except for some minor discrepancies near the hub. The uniform temperature ratio profile obtained from through-flow simulations by imposing a uniform viscous coefficient at the TE is patently unrealistic and incapable of capturing local viscous phenomena. Due to lack of flow mixing inside the volute (shear terms in through-flow code were deactivated), the temperature profile undergoes subtle changes in its shape as the flow moves towards the outlet. The profiles of the aforementioned quantities derived from CFD simulations show the presence of extensive mixing in the volute, leading to a uniform temperature ratio profile at the outlet of the volute which is very well matched by the simple uniform coefficient.

This hypothesis is further verified by comparing the uniformity of the pressure ratio, temperature ratio and efficiency CFD profiles at the outlet which are approximated more precisely by the simple, uniform distribution coefficient (figure(5.51)). This highlights the importance of spanwise mixing in through-flow codes, especially at the presence of large ducted domains. In the absence of explicit flow mixing, e.g. by introducing shear stresses in the governing equations, the inclusion of uniform distribution coefficients at the last stage, whenever a long ducts are present downstream, seems to provide a reasonable and less complex modelling alternative solution to the spanwise mixing approach.

Figures(5.52) and (5.53) illustrate the distribution of the stagnation properties as well as the static pressure and meridional Mach number along the duct of CSTAR

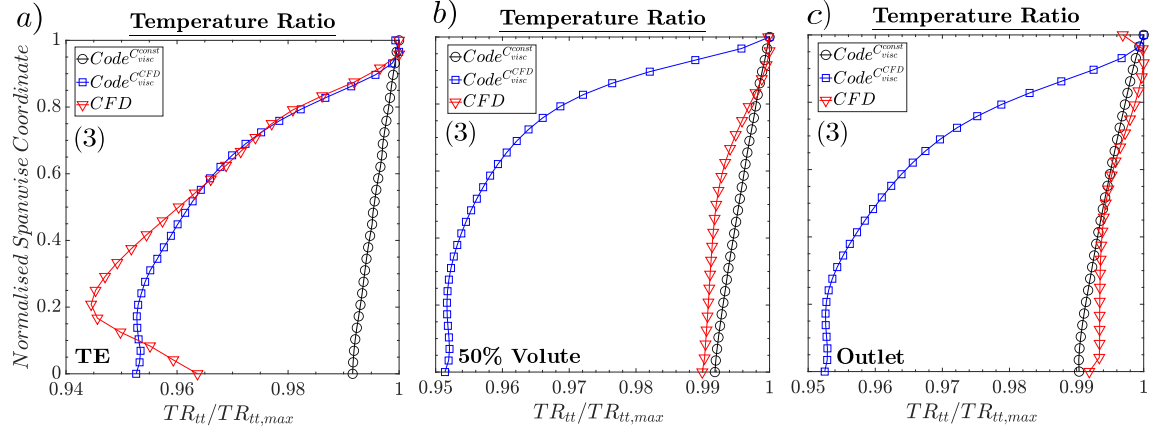


Figure 5.50: Variations of total temperature ratio profile at different locations downstream the vaned diffuser obtained by CFD simulations and the code using CFD and uniform coefficients.

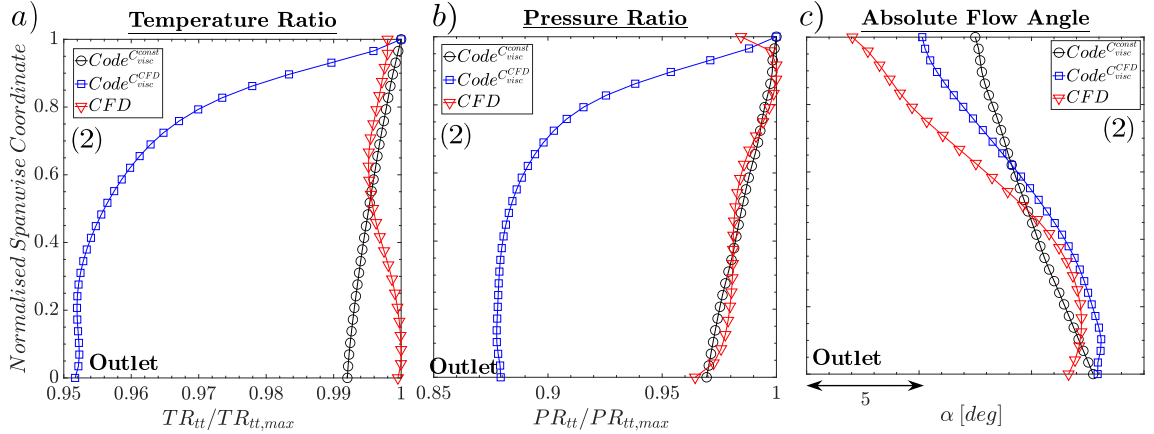


Figure 5.51: Comparison of a) total temperature ratio, b) total pressure ratio and c) flow angle spanwise distribution at the outlet obtained by CFD simulations and the code using CFD and uniform coefficients.

compressor. The total pressure and temperature contours of the code are in very good agreement with the CFD results. The most notable difference is the higher stagnation pressure and temperature regions at the shroud, downstream the impeller TE along with a noteworthy pressure drop at the hub of the vaned diffuser passage (figure(5.52b)). The latter is related to the presence of higher entropy-generation phenomena which are underestimated by the viscous force distribution coefficient used in the vaned diffuser.

The static pressure contours derived from body force simulations, shown in figure(5.53a), are in good agreement with the CFD contours. Both the pressure distribution along the duct, as well as the static pressure rise is predicted exceptionally well. Dissimilarities in the meridional Mach number contours are primarily observed near the shroud of the impeller and downstream the TE of the impeller. The first is

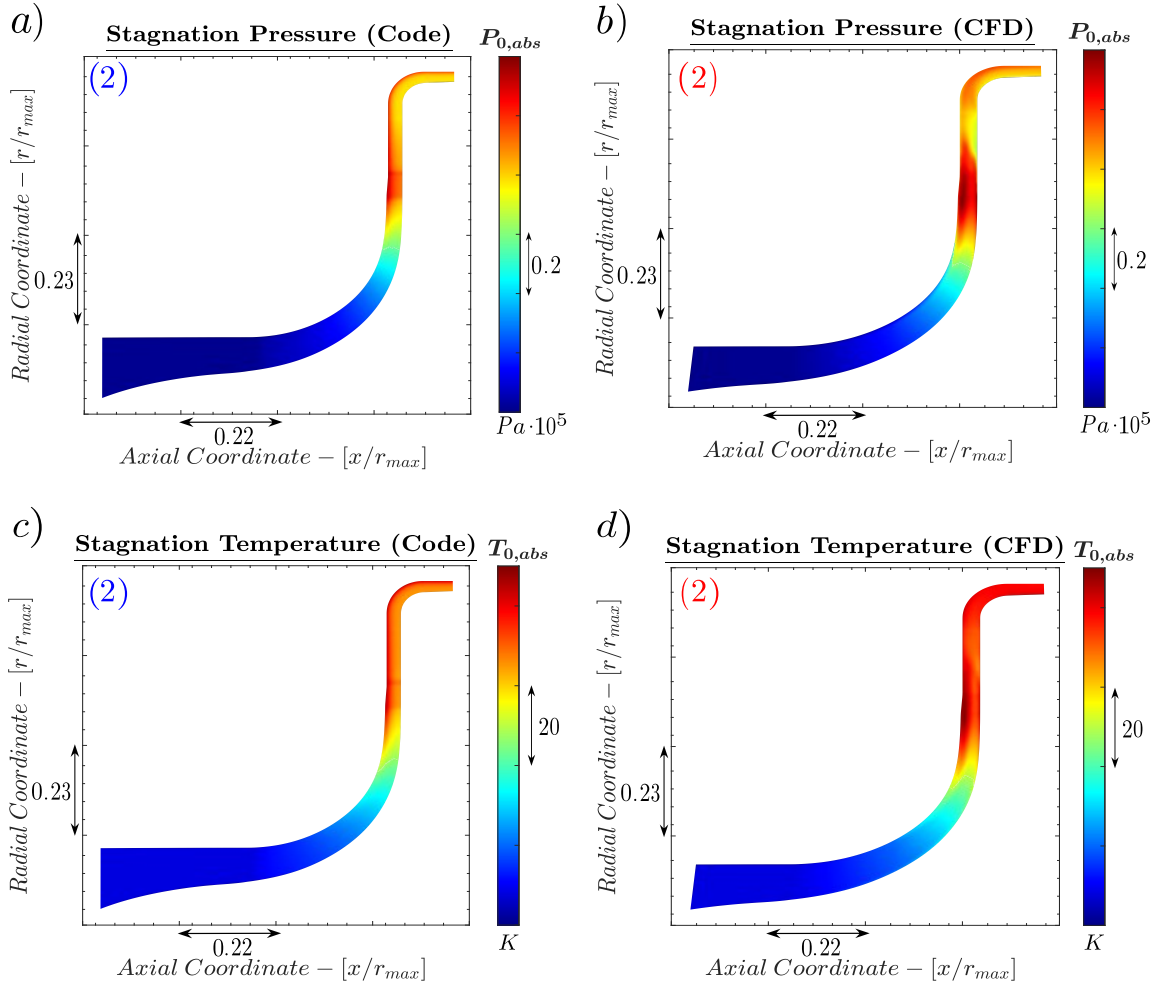


Figure 5.52: Comparison of CSTAR compressor stagnation pressure and temperature contours obtained by the through-flow code and the CFD model.

related to viscous phenomena and to the tip clearance gap, causing flow separation at the shroud. Tip clearance gap phenomena cannot be modelled in the code and are only considered in the analysis by the loss correlations via an increase in the predicted average pressure loss. The propagation and mixing-out of that separation region downstream the impeller alters the flow-field. Low Mach number regions near the hub and the shroud in the volute can also be seen in figure(5.53d) and are related to boundary layer growth and flow separation. The deceleration of the flow and the boundary layer in the diffuser enhances boundary layer growth which eventually results in flow separation near the hub and the shroud. Even though local, inherently viscous or non-axisymmetric phenomena cannot be reproduced by the through-flow code e.g. separated flow at the shroud, boundary layers, tip leakage vortices etc., the overall flow-field structure is captured reasonably well.

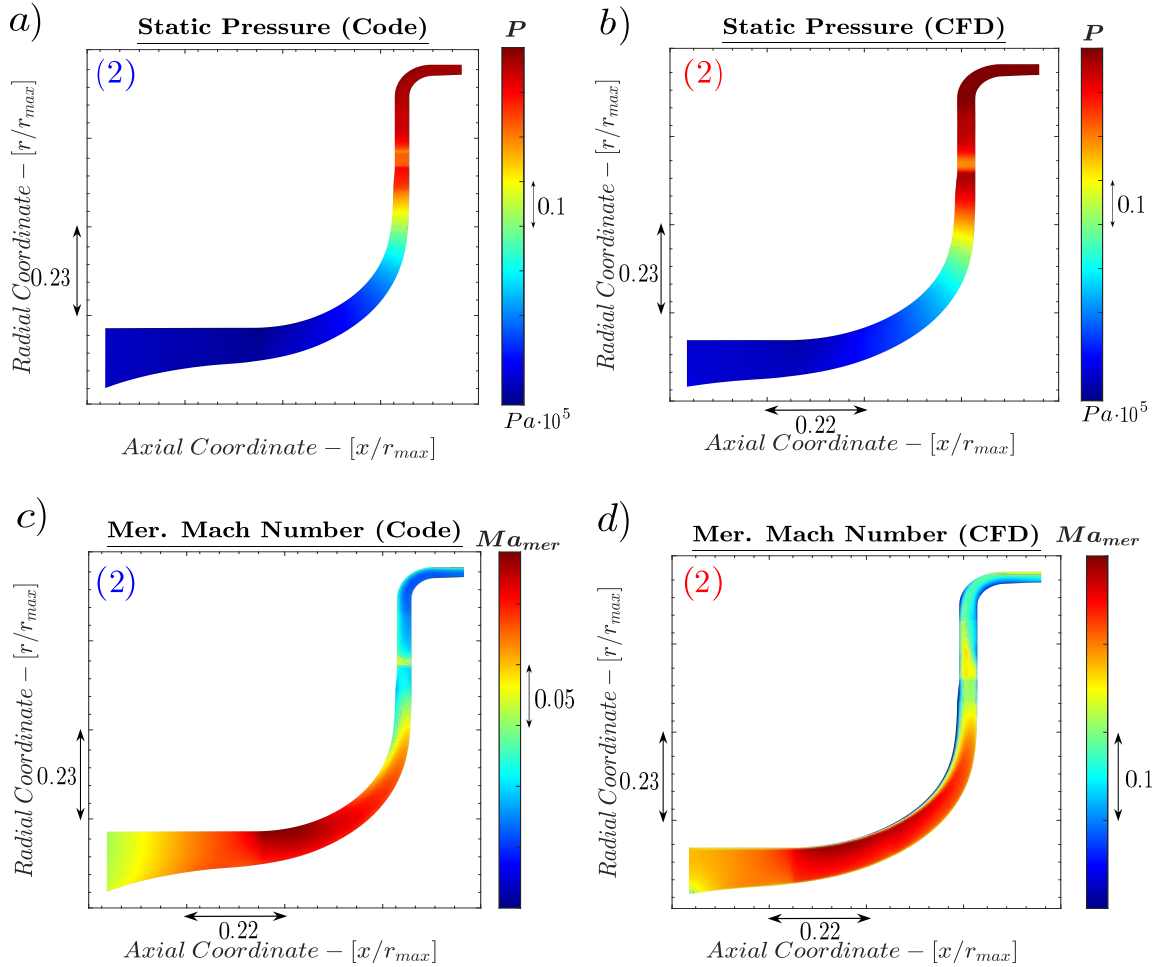


Figure 5.53: Comparison of CSTAR compressor static pressure and meridional Mach number contours obtained by the through-flow code and the CFD model.

5.2 Reverse flow

In the previous section, the through-flow code was validated against experimental and CFD data. It was demonstrated that through-flow simulations are able of predicting global, aerodynamic performance of axial and centrifugal compressors, while being capable of reconstructing the flow-field with reasonable accuracy. The limitations of the developed framework are more pronounced whenever strong viscous-related and potentially 3D phenomena manifest e.g. tip leakage vortices, flow separation etc., which cannot be accurately reconstructed via the distribution coefficient method nor can be replicated by an axisymmetric solver. The extraction of the viscous force coefficient is subject to the limitations of the viscous force implementation method, while the extraction of the deviation coefficient method is mostly subject to interpolation and blade curvature errors.

At reverse flow conditions, viscous-induced phenomena are expected to dominate the formation of essential flow-field features such as highly separated flows, aerodynamic blockage, vortices and mixing. While a precise replication of those flow-features is outside the capabilities of Euler-based, through-flow codes, the implications of those phenomena on global aerodynamic performance predictions is substantial. To consider the effect of those phenomena in the analysis, explicit, reduced-order modelling approaches are incorporated. Their validity and limitations were discussed in (4.6).

The verification of the overall developed reverse modelling approach is performed in a similar way; steady-state reverse flow characteristics and flow-field contours obtained from CFD simulations are compared against the through-flow code ones. The steady-state reverse flow verification is carried out exclusively against CFD data due to absence of experimental reverse flow characteristics in the public domain. While steady-state reverse flow characteristics are obtained by increasing the static pressure at the outlet, transient, post-stall simulations require the inclusion a numerical plenum-throttle model which provides boundary conditions at the outlet. The throttle is initially closed, increasing the static pressure in the volume and forcing the compressor to stall. Once the compressor stalls and flow reversal manifests, the compressor surges; the throttle is then opened to allow the compressor to recover. Surge cycles of the NASA CC3 compressor with vaned and vaneless diffuser and different plenum volumes are obtained. Post-stall validation is performed against experimental data of the NASA CC3 compressor with vaned diffuser.

5.2.1 Steady-state aerodynamic performance

The capability of the proposed stalled flow modelling approach to predict the steady-state aerodynamic behaviour of centrifugal compressors is assessed on the basis of 4 quantities which represent essential compressor aerodynamic performance parameters; total-to-total and total-to-static pressure ratio as well as torque and torque parameter, with the latter defined as the torque normalised by the total pressure at the flow inlet (geometric outlet at reverse flow conditions). The torque provides a more intuitive metric to quantify the energy exchange between the compressor and the flow at reverse flow conditions. The torque parameter relates the torque with the inlet total pressure associated with the operating point under consideration. The aforementioned quantities are derived using steady-state, single-passage, RANS simulations of three centrifugal compressor geometries; the NASA CC3 compressor

with vaned and vaneless diffuser and the impeller SRV2. Significant numerical stability issues were encountered with the CSTAR compressor geometry, preventing the acquisition of reliable reverse flow characteristics, thus reverse flow results are not shown here. Verification of reverse flow modelling is restricted to centrifugal compressors since the axial compressor reverse flow modelling approach of ACroSS has been adopted, which has been validated for low-speed and high-speed multi-stage axial-flow compressors. The geometries and the corresponding CFD models were presented in detail in section(5.1). The boundary conditions for both the CFD and the through-flow simulations are switched to open B.C.s, allowing the flow to enter and exit the domain boundaries depending on the direction of the flow. The global aerodynamic performance quantities are plotted against corrected mass flow, which is the mass flow corrected using flow properties at the geometrical outlet of the compressor (flow inlet at reverse flow conditions). The total-to-total pressure characteristics are obtained using area-averaged quantities since it was shown to provide smoother characteristics while being less sensitive to the large recirculation regions at the inlet duct.

DLR SRV2-0 - Reverse flow

The first test case for the verification of the reverse flow modelling approach, is performed on the SRV2-0 compressor geometry. The length of the modified curvature region, is set to 10% and the mixing parameter to 0.4. Figure(5.54) shows the characteristics of the aforementioned quantities at 40%, 60% and 80% speed. The design speed reverse flow characteristic is not included in the graphs, since convergence was not deemed satisfactory. The pressure ratio predictions of the through-flow code are in good agreement with the CFD results but are consistently underestimated compared to the CFD simulations. The shape of the characteristics obtained by the code are consistent with the CFD curves, exhibiting similar trends and accurately predicting the choking mass flow. The through-flow code predictions correlate better in the lower NRT range and deviate more from CFD results in the higher NRT range. The total-to-total pressure ratio differences are between 8.5% - 9.5% in the mid-mass flow region and are less than 4.5% in the highest and lowest mass flow region on the 40% and 60% characteristics. The discrepancies at the 80% characteristic are consistently within the [9%-10%] range. The total-to-static pressure ratio differences are of the same order. The higher discrepancies at 80% speed are primarily related to less energy input of the through flow-code compared to CFD and secondarily to overprediction of the mass flow by the through-flow code which moves the characteristic

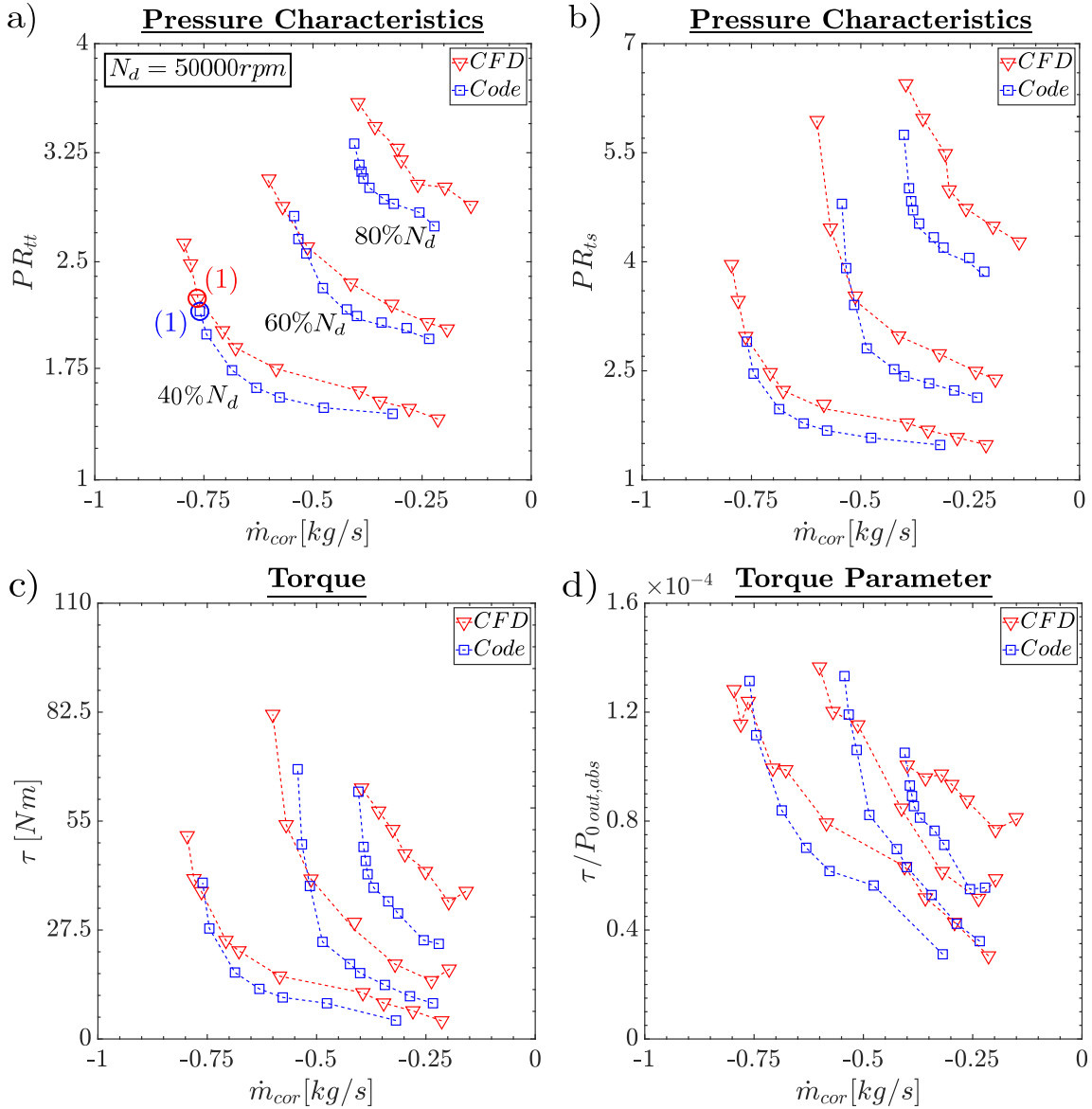


Figure 5.54: Comparison of a) total-to-total pressure ratio and b) total-to-static pressure ratio c) torque and d) torque parameter reverse flow characteristics of compressor SRV2-0.

towards the left. The torque and torque parameter curves confirm that less energy is imparted into the flow by the through flow-code compared to the CFD, which explains the higher pressure rise predictions of the CFD simulations. The discrepancies in the torque predictions are similar to the pressure ratio ones and are in the range [2% - 10%]. The higher energy input of the CFD model with respect to both the experimental data and the through-flow code predictions was also observed in forward flow operation. This was amplified at higher rotational speeds and is also reflected on the reverse flow characteristics, thus the lower torque predictions of the code are expected to be more realistic.

During reverse flow operation, the flow-field is characterised by non-uniformity

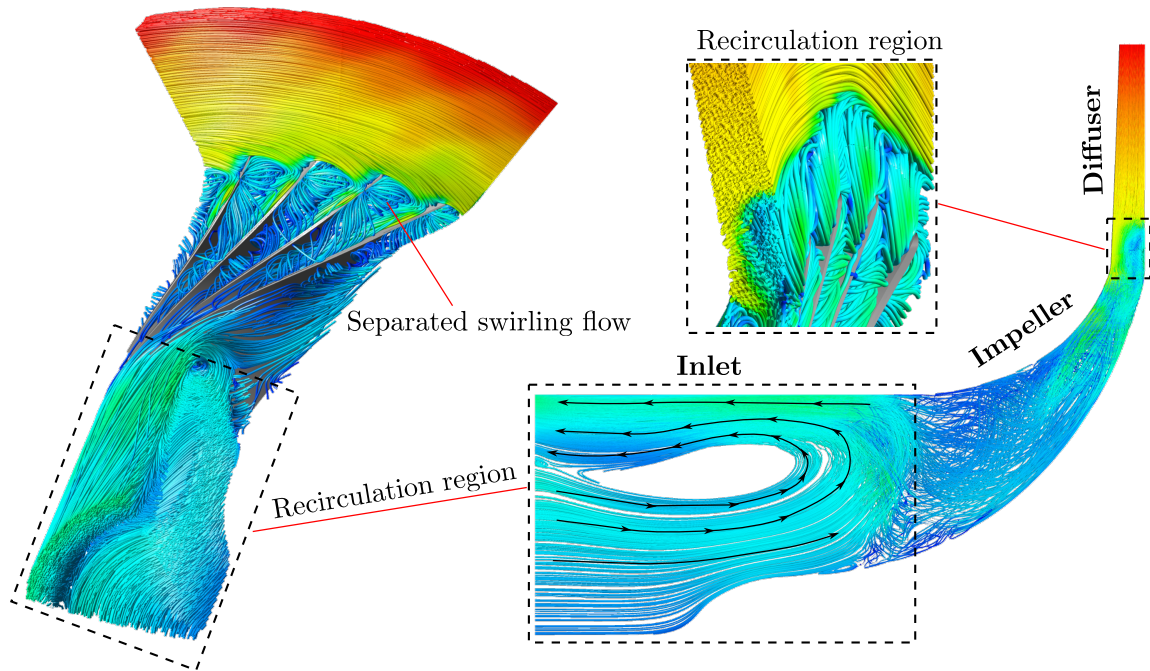


Figure 5.55: Three-dimensional streamlines in SRV2-0 compressor, illustrating the recirculation regions at the inlet domain and near the TE obtained from steady-state, single-pass RANS simulations.

in both the meridional and circumferential direction as well as complex, three-dimensional flow features. All tested centrifugal compressor geometries exhibit several common flow features; a recirculation region at the inlet domain, a recirculation region near the TE hub, highly separated, swirling flow in the blade passages and shock waves at the TE. Figure(5.55) shows the 3D streamlines obtained from steady-state, single-pass RANS simulations at 60% speed and low mass-flow/low pressure ratio operation. A large recirculation region is forming at the inlet bell mouth, with the flow at the hub moving in the positive direction and the flow near the shroud towards the negative direction; this restricts the flow exiting from the hub, shifting the streamlines towards the shroud and forcing the flow to exit the impeller through a stream near the shroud. Near the TE hub, a recirculation region is also forming due to the interaction of the impeller with the incoming flow, impeding it from entering the passage. This effect gradually subsides in higher spanwise positions, allowing the flow to enter the impeller through the shroud.

The presence of the recirculation region reduces the velocity at the TE near the hub (figure(5.56a)) but the associated aerodynamic blockage results in high acceleration of the flow in higher spanwise positions. The reduction in the local blade pitch due to the presence of the splitter blade combined with the reduced effective area results in the formation of shock waves at the TE in higher spanwise positions

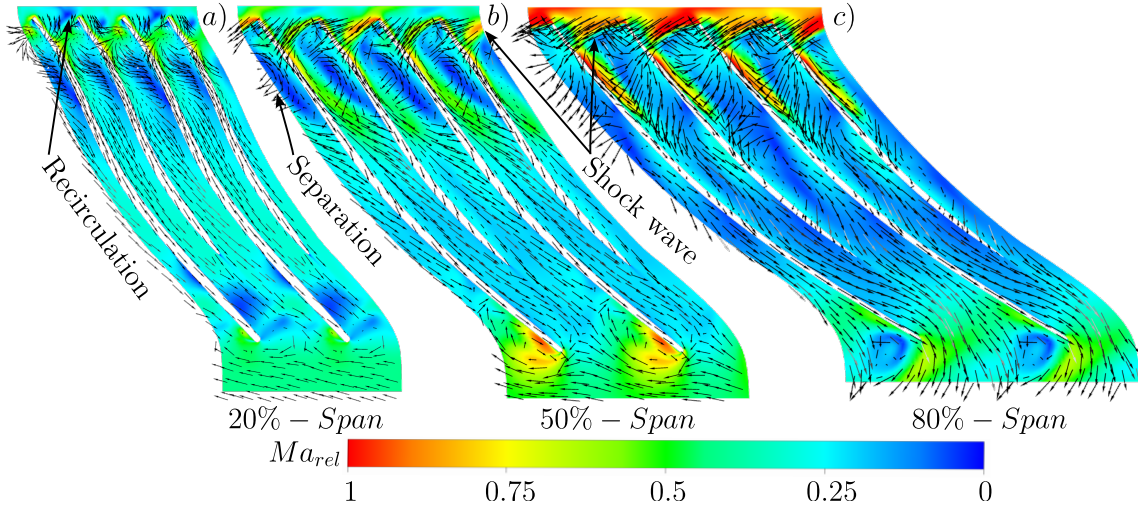


Figure 5.56: Relative Mach number contours and velocity vectors along SRV2-0 impeller passage obtained from CFD simulations, showing regions of separated flow, recirculation and shock waves during reverse flow operation near the a) hub, b) midspan and c) shroud.

(figures(5.56b,c)). Low-momentum regions extending downstream the TE (towards the LE), indicating flow separation, are present at all spanwise positions with the mixing process taking place primarily inside the passage.

To evaluate the accuracy of the flow-field solution obtained with the through-flow code, the streamwise velocity and static pressure contours are compared against contours derived from 3D, steady-state, RANS simulations. The contours shown in figure(5.57) correspond to the highest PR setting of the 40% characteristic. The static pressure distributions at the inlet and outlet domains are replicated exceptionally well by the through-flow code. In the impeller passage, the most notable discrepancies are primarily located in the impeller TE, where a significant reduction in static pressure occurs further downstream the TE of the impeller in the through-flow code compared to the CFD simulations. This is caused by an excessive acceleration upstream the TE (figure(5.57)d) due to the presence of the recirculation region at the hub which generates blockage and is not reproduced by the through-flow code. Inside the impeller passage, near the hub, a substantially more extended acceleration region is predicted by the though-flow code near the hub, exhibiting a more non-uniform pattern composed of two distinct, higher and lower velocity streams. This is related to the swirling of the flow in the relative FoR, which is moving with a rather uniform meridional velocity towards the LE, a flow feature which can only be captured if distinct blade passages are modelled. The swirl of the flow in the blade passage also enhances flow mixing, the intensity of which, is underestimated by the artificial viscosity terms; the intensity of the mixing is adjusted by the artificial

viscosity coefficient which is set to a low value to prevent numerical instabilities. The streamwise velocity distribution at the inlet bell-mouth is accurately reproduced by the through-flow code, predicting both the size of the reverse flow stream from which the flow exits the blade, as well as the size of the forward flow region.

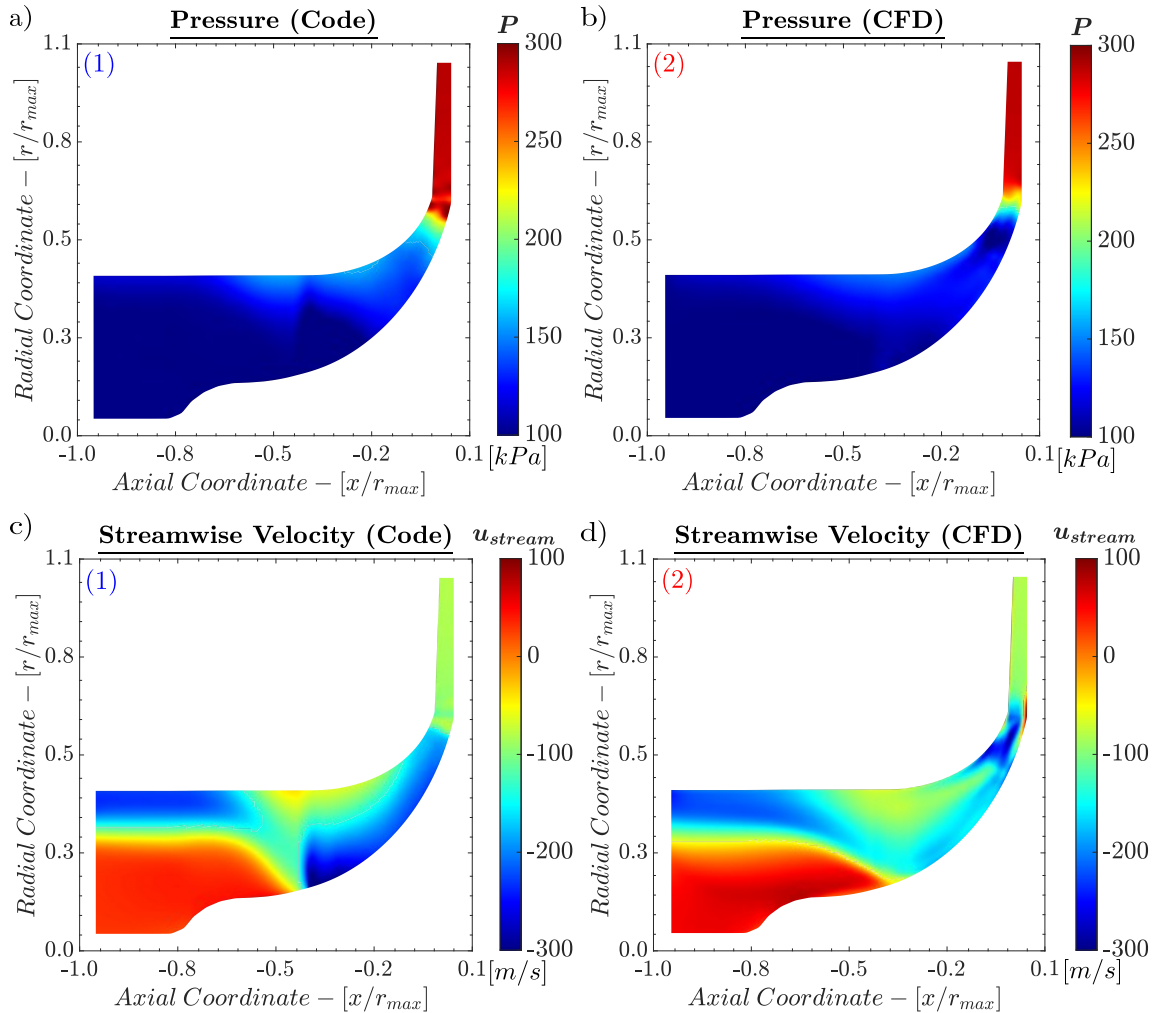


Figure 5.57: Static pressure and streamwise velocity contours comparison of SRV2-0 compressor with vaneless diffuser obtained by the through-flow code and the CFD model at steady-state reverse flow conditions and 40% speed.

NASA CC3 with vaneless diffuser - Reverse flow

Reverse flow simulations are subsequently performed on the NASA CC3 compressor with vaneless diffuser since it essentially is an identical configuration to that of SRV2-0. The modified curvature region is set to 40% and the mixing parameter to 0.3 for all characteristics since it was found to yield more accurate total pressure ratio predictions. The global aerodynamic performance quantities are extracted from steady-state through-flow and CFD flow-field solutions at 40%, 60%, 80% and 100%

speed (figure(5.58)). The design speed and the corrected mass flow are corrected using flow properties at the diffuser geometrical outlet.

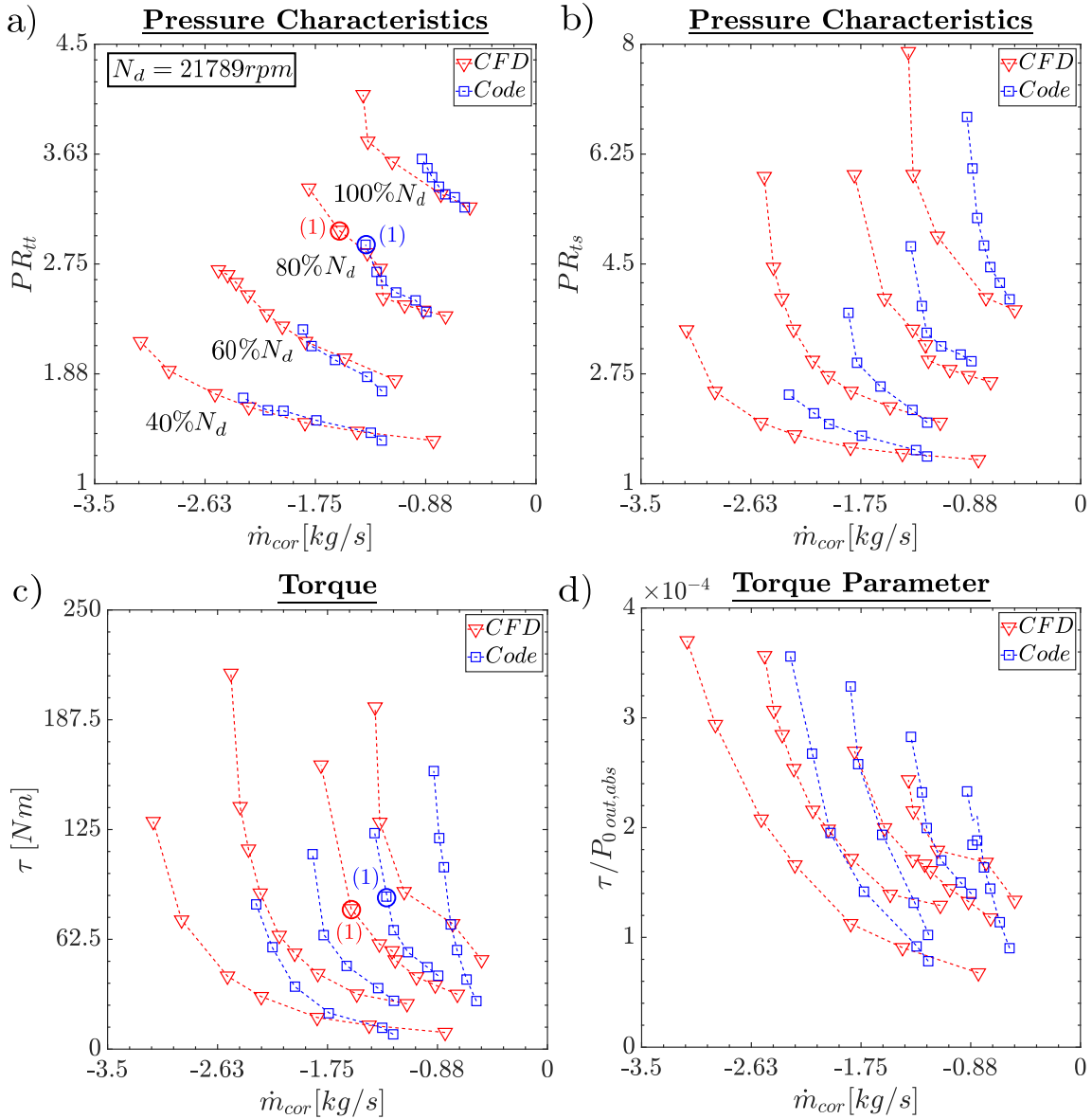


Figure 5.58: Comparison of a) total-to-total pressure ratio and b) total-to-static pressure ratio c) torque and d) torque parameter reverse flow characteristics of compressor CC3 with vaneless diffuser.

The total-to-total pressure ratio characteristics obtained with both models correlate exceptionally well in the 60%-80% speed range. The discrepancies between the two models are more pronounced at design speed, where the steeper gradient of the through-flow simulations indicate an overestimation of the blockage which results in lower choking mass flow. The total-to-static pressure ratio characteristics (figure(5.58b)) correlate well with the CFD data in the lower mass flow region but the code tends to increasingly overpredict pressure ratios at higher mass-flows which

presumably is the outcome of the combined effect of primarily the overestimated aerodynamic blockage and secondarily of higher flow turning. The torque and torque parameter curves exhibit identical trends and are very consistent with the CFD results but are shifted towards the lower mass flow region, further confirming the hypothesis that the source of errors is predominantly related to overestimated aerodynamic blockage predictions. This could be addressed by either increasing the extent of the modified curvature region and/or by increasing the mixing rate in the aerodynamic blockage model. Adjustments were avoided since it was preferred to perform simulations of both CC3 configurations (vaned and vaneless diffuser) with the same settings.

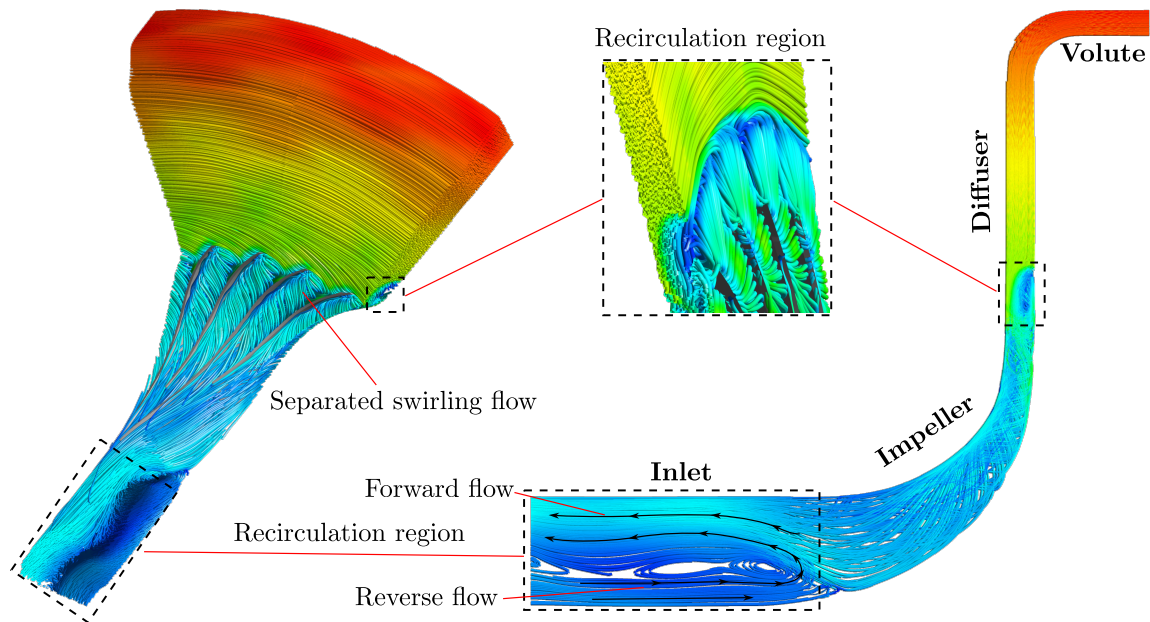


Figure 5.59: Three-dimensional streamlines in NASA CC3 compressor passage with vaneless diffuser, illustrating the recirculation regions at the inlet domain and near the TE obtained from steady-state, single-passages RANS simulations.

An overview of the flow-field, visualised using 3D streamlines in the relative FoR along with important flow features is shown in figure(5.59). The flow features highlighted earlier are also exhibited here; an extensive and a localized circulation region form at the inlet domain and near the TE hub respectively, while a highly swirling separated flow is dominating the blade passage. The size of the recirculation regions is governed by the pressure gradient along the duct; larger pressure gradients allow the higher momentum-flow entering and exiting the passage to suppress the recirculation regions. Similar to impeller SRV2-0, the recirculation region near the TE gradually diminishes towards the shroud.

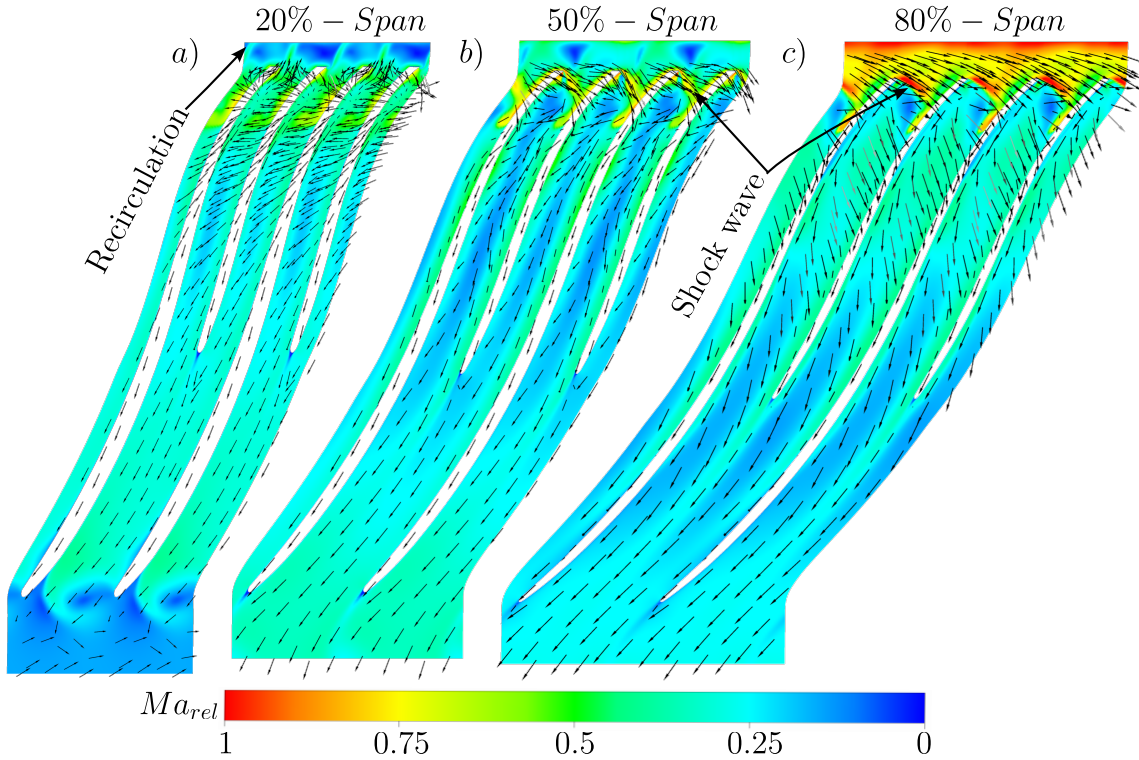


Figure 5.60: Relative Mach number contours and velocity vectors along impeller passage of NASA CC3 compressor with vaneless diffuser obtained from CFD simulations, showing regions of separated flow, recirculation and shock waves during reverse flow operation near the a) hub, b) midspan and c) shroud.

Velocity vectors and relative Mach number contours at 20%, 50% and 80% span obtained from the CFD flow-field solution are depicted in figure(5.60). The blockage precipitated by the interaction between the impeller and the incoming flow, substantially decelerates the flow closer to the hub which in turn, results in a less separated, more uniform flow near the hub. At midspan and 80% span, large, low-momentum flows surrounded by higher velocity streams manifest. The high velocity regions essentially are the traces of the high-momentum flow swirling in-between the pressure and suction side of the full and splitter blade respectively (see figure(5.59)). The relative swirl velocity component gradually diminishes, the flow aligns with the blade and mixes-out as it approaches the LE. The formation of shock waves at the higher spanwise positions near the TE is also captured by the Mach number contours.

The validity of the flow-field solution is further examined qualitatively by comparing static pressure and streamwise velocity plots. The contours correspond to the operating point shown in figure(5.58) on the 80% characteristic and was selected on the basis of the imposed boundary conditions and torque outputs, which are almost identical. Both the pressure and the streamwise velocity contours of the code are in

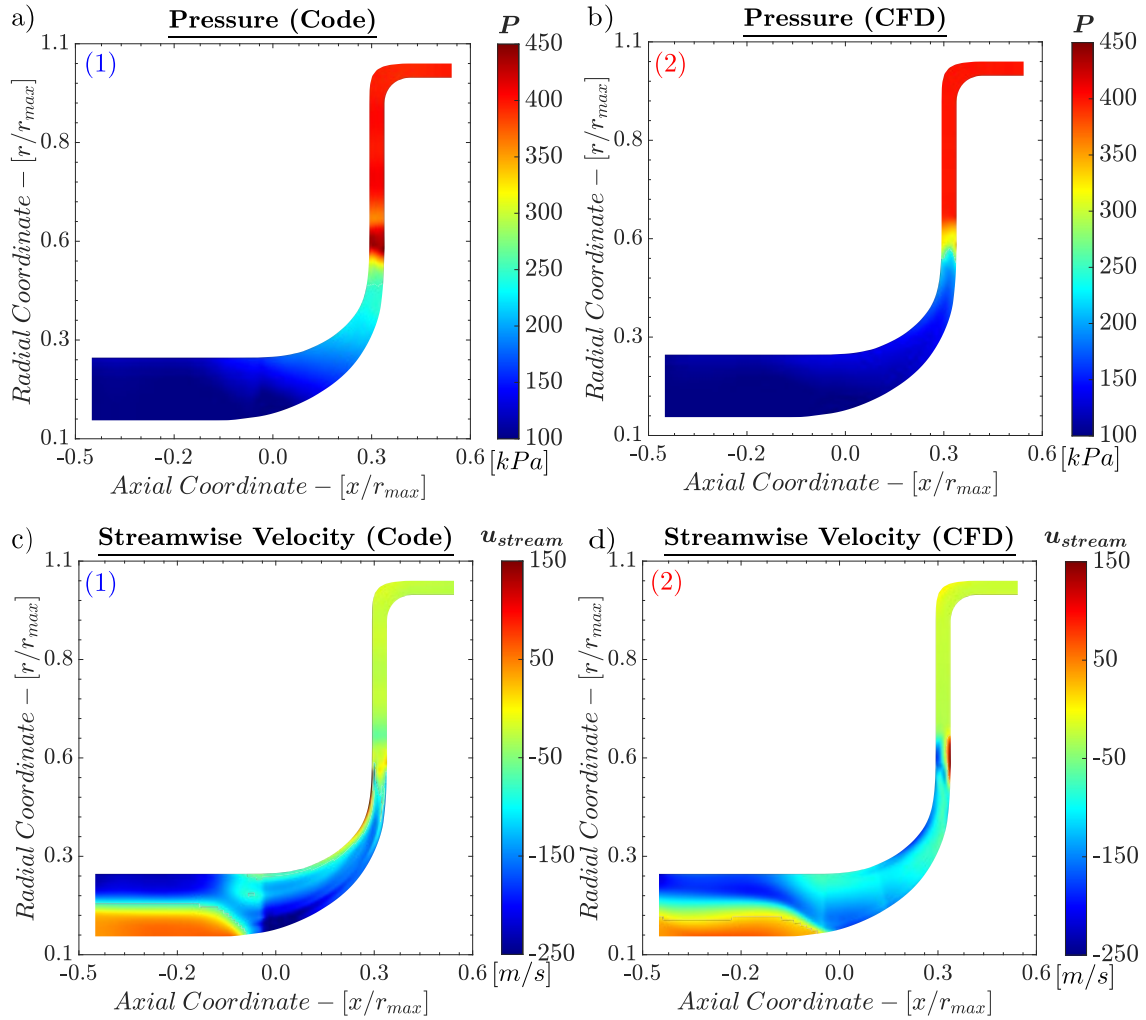


Figure 5.61: Static pressure and streamwise velocity contours comparison of CC3 compressor with vaneless diffuser obtained by the through-flow code and the CFD model at steady-state reverse flow conditions and 80% speed.

good agreement with the CFD ones. The static pressure contours of the through-flow code show the presence of an acceleration region which extends from the TE up to approximately the middle of the blade passage in the streamwise direction. In the CFD simulations, this acceleration occurs more abruptly, near the TE and the vaneless space region. The streamwise velocity distributions (figure(5.61c,d)) show that the code predicts in general, higher velocities inside the blade passage but slightly lower towards the shroud compared to the CFD ones. This is linked to mixing effects and the swirl of the flow in the relative FoR inside blade passage, resulting in a more uniform distribution of the streamwise velocity on the meridional plane. The pressure and velocity distributions at the inlet and the outlet correlate exceptionally well, showing that the code is capable of accurately predicting the extend of both the forward and reverse flow streams at the inlet region.

NASA CC3 with vaned diffuser- Reverse flow

The final test case is the NASA-CC3 compressor with vaned diffuser. The CFD model comprises a shorter inlet compared to the vaneless diffuser model. The through-flow simulations are performed using an identical computational model to the one used for the vaneless diffuser simulations by applying a body-force field in the vaned diffuser region and imposing the corresponding blade metal blockage. The extent of the modified curvature region is set to 40%, equal to the one used in the vaneless diffuser configuration. The pressure and torque characteristics corresponding to body force and CFD simulations are depicted in figure(5.62).

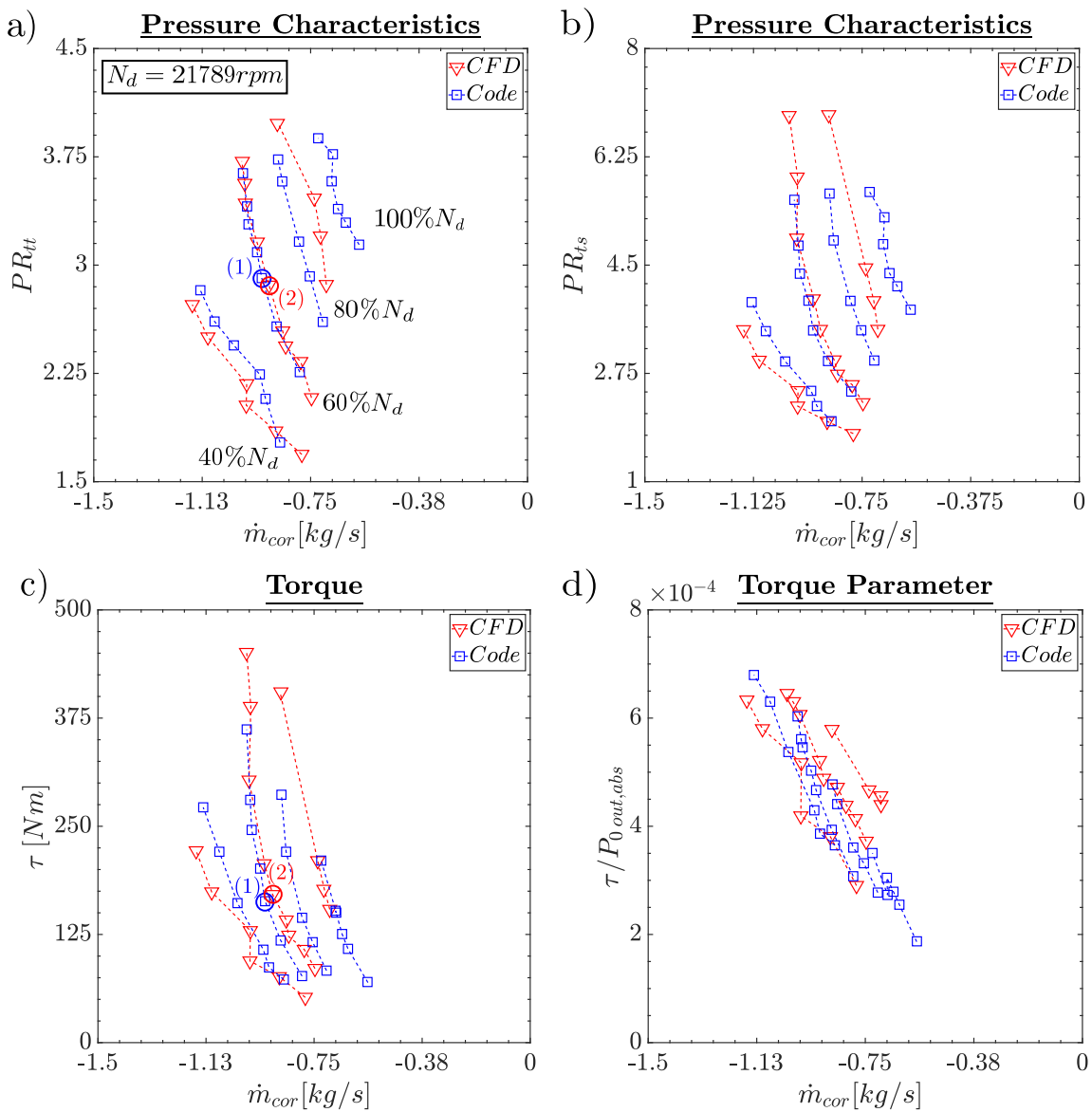


Figure 5.62: Comparison of a) total-to-total pressure ratio and b) total-to-static pressure ratio c) torque and d) torque parameter reverse flow characteristics of compressor CC3 with vaned diffuser.

The derivation of reliable, steady-state operating points at 100% design speed was particularly challenging due to the large oscillations in the mass flow and pressure ratio and are not included here. Similar issues were encountered at 80% design speed, nevertheless four stable operating points were extracted and are included in the dataset. The total-to-total and total-to-static pressure characteristics obtained by the code are in very good agreement with the CFD results along the entire speed range both in terms of mass-flow capacity and pressure rise. At 40% speed, the pressure rise is slightly overestimated by the code, while at 80% speed the mass flow capacity is slightly underpredicted. The torque predictions also correlate exceptionally well with the CFD predictions in the lower 40% and 60% speed range and are slightly underestimated at 80% design speed. This indicates that flow turning and consequently the energy exchange between the flow and the blades is accurately introduced via the body-force model.

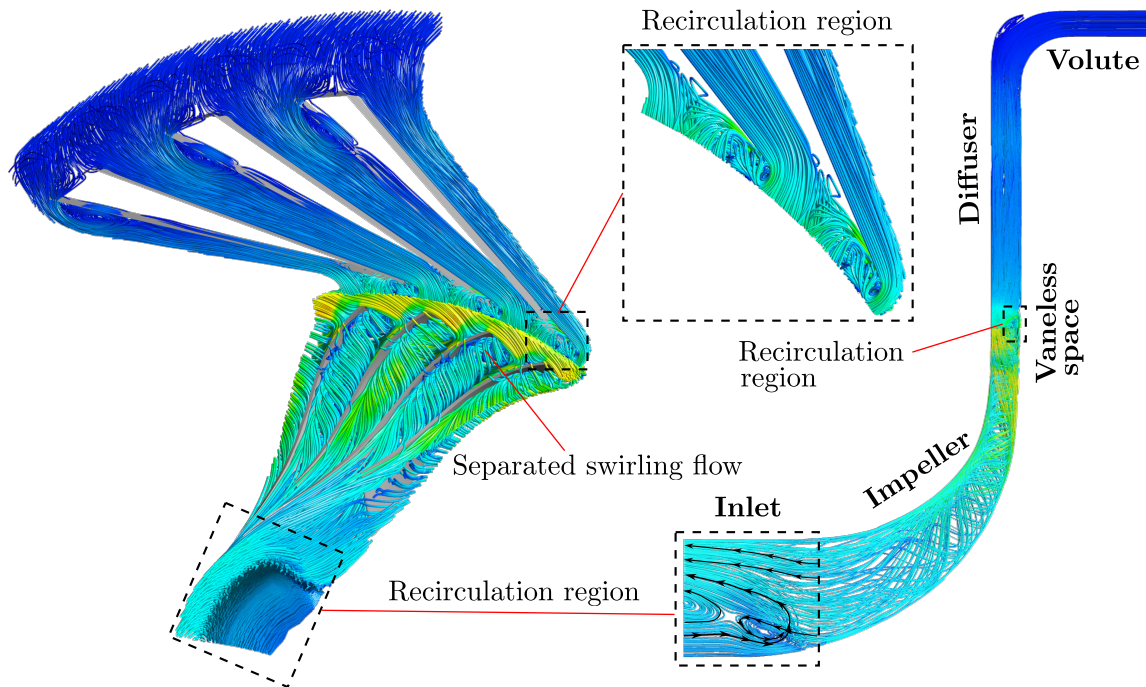


Figure 5.63: Three-dimensional streamlines in NASA CC3 compressor with vaned diffuser, illustrating the recirculation regions at the inlet domain and near the TE obtained from steady-state, single-passage RANS simulations.

An overview of the flow-field during reverse flow operation of the NASA CC3 compressor with vaned diffuser is illustrated in figure(5.63). The inlet domain of the CFD model is less elongated, thus the recirculation region is less extended, but exhibits the same behaviour with the SRV2 compressor and the vaneless configuration; its size is sensitive to the pressure gradient in the duct. At the diffuser blade LE, vortical structures are convected from the impeller back to the vaneless space, reaching

the inlet and causing the streamlines, especially near the shroud to diverge from the camberline. The flow in the impeller is identical to the vaneless diffuser configuration and is characterised by large regions of separated, swirling flow in the relative FoR. As the flow moves downstream the LE towards the TE, the relative swirl velocity component diminishes and the flow becomes more uniform, exiting the blade passage at the LE from higher spanwise due to blockage introduced by the recirculation region at the inlet duct.

Since the flow characteristics inside the impeller do not exhibit any major, noteworthy differences from the ones observed in the vaneless configuration, only the velocity vectors and Mach number contours of the diffuser passage are shown in figure(5.64). At all three spanwise positions, the presence of a separation region at the suction side, near the TE can be observed. This is caused by the high incidence angle of the incoming flow and the flat, cut-off trailing edge of the blades. The influence of the vortical structures convected back from the impeller is reflected on the non-uniform Mach number distributions at the LE of the blades. Near the hub, their impact is mainly limited at the suction side, in the vicinity of the LE and is not interfering substantially with the bulk flow exiting the passage. At higher spanwise positions, the presence of those vortical structures is affecting a larger portion of the inlet blade passage causing the flow to separate. This was also discussed in section(4.6).

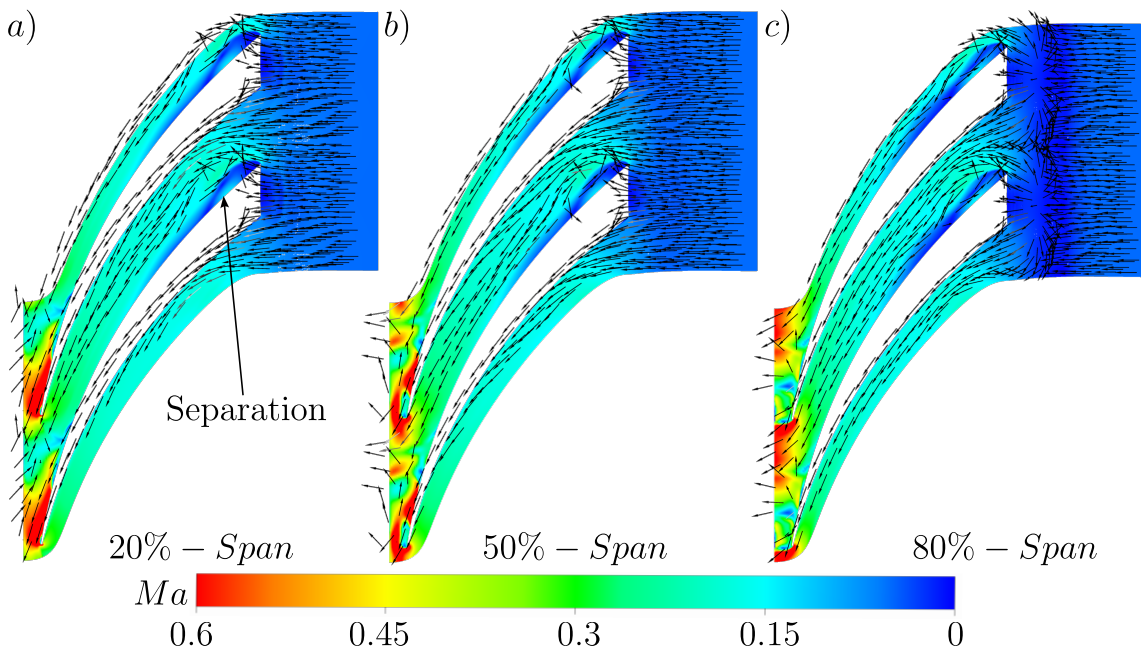


Figure 5.64: Absolute Mach number contours and velocity vectors along impeller passage of NASA CC3 compressor with vaneless diffuser obtained from CFD simulations, showing regions of separated flow, recirculation and shock waves during reverse flow operation near the a) hub, b) midspan and c) shroud.

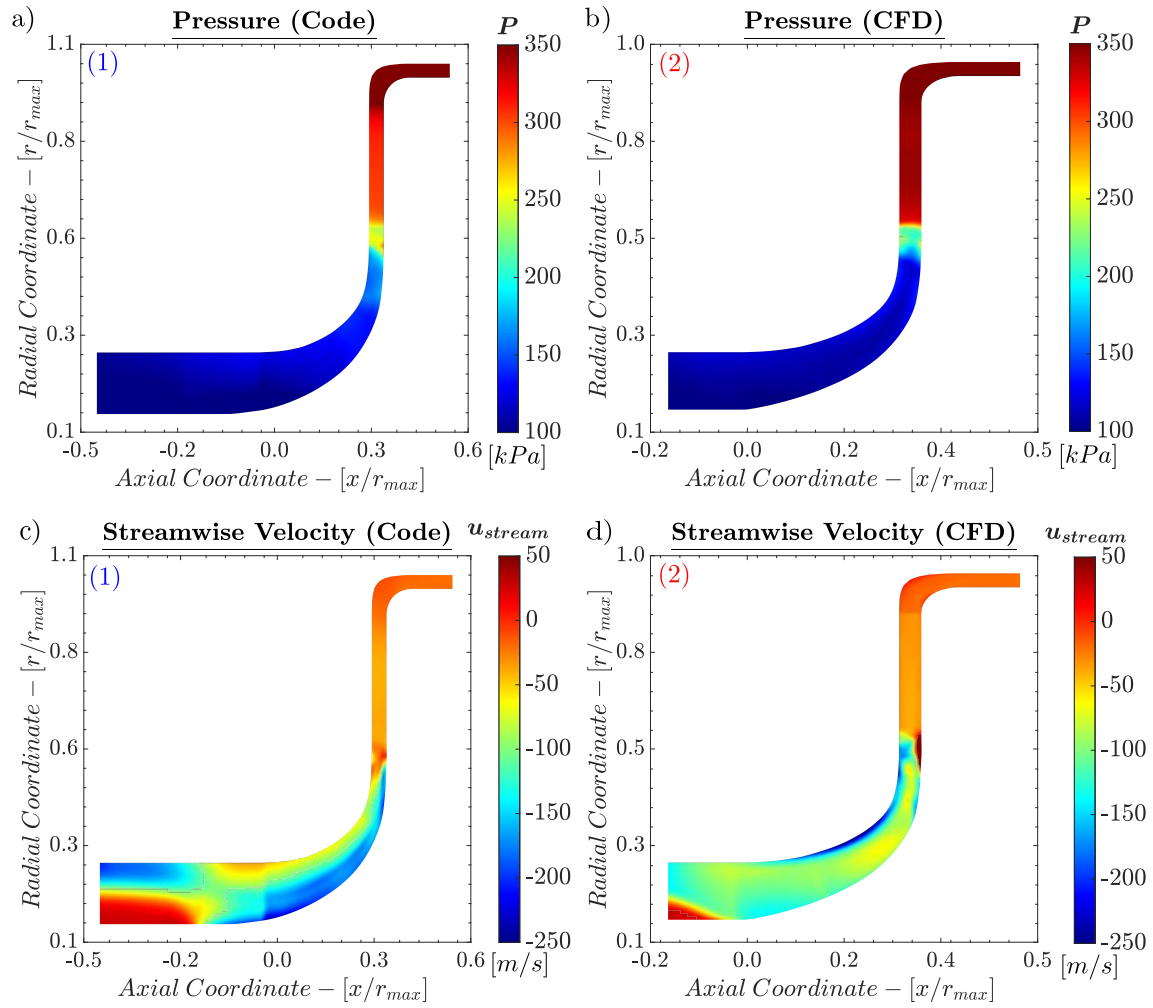


Figure 5.65: Static pressure and streamwise velocity contours comparison of CC3 compressor with vaned diffuser obtained by the through-flow code and the CFD model at steady-state reverse flow conditions and 60% speed.

The pressure and streamwise velocity contours shown in figure(5.65) correspond to the operating point on the 60% characteristic, shown in figure(5.62). The static pressure contours of the through-flow code in the vaned diffuser passage show a noteworthy reduction in the static pressure which is not captured by the CFD simulations. The density contours (not included here) showed an analogous distribution, while the temperature distributions are in agreement with the CFD ones; since the pressure is a function of the density and the temperature (ideal gas law) it can be deduced that the discrepancy is related to the density. The streamwise velocity in the vaned diffuser passage is in very good agreement with the CFD results therefore the discrepancy cannot be attributed to overestimation of aerodynamic and/or metal blockage. Apart from the vaned diffuser passage, the pressure distribution along the duct is in very good agreement with the CFD one. The streamwise velocity contours

in the impeller passage obtained by the code exhibit similar trends to the vaneless diffuser configuration; near the hub the flow accelerates excessively, but gradually decelerates towards the shroud, in the spanwise direction. The CFD contours show a more mixed-out, uniform distribution with a high velocity region near the shroud. As mentioned in the previous section, these are related to enhanced mixing and flow swirl in the relative FoR, and cannot be accurately captured by the mixing model and the absence of discrete blade passages in the through-flow simulations.

5.2.2 Transient aerodynamic performance

The transient, post-stall aerodynamic response of the two NASA CC3 configurations comprising vaneless and vaned diffusers is simulated with the through-flow code. Different surge cycles are obtained at design speed by varying the size of the plenum. Experimental surge data are available only for the configuration with the vaned diffuser and are used to validate the code [61]. The experimental data comprise static pressure measurements at the impeller inlet, vaneless space, vaned diffuser inlet and throat including DPIV measurements at the diffuser. Mass flow, pressure ratio or torque trajectories of the surge cycle are not provided. The coordinates of the modified inlet duct, the size of the plenum and details on the ducting configuration are not reported either. The 1D plenum-throttle numerical model, presented in section (3.8), is used to replicate the presence of the plenum volume and enable the dynamic behaviour of the boundary conditions during surge. The throttle area is initially closed, pressurizing the plenum and shifting the operating point of the compressor towards the surge line. Once the compressor surges and reverse flow establishes in the compressor, the throttle is fully opened to allow the compressor to recover. Open boundary conditions are imposed, allowing the flow to enter and exit at the boundaries. The post-stall behaviour of the compressors is investigated at different plenum volumes: $1m^3$, $3m^3$, $5m^3$ and $7.5 m^3$. The experimentally reported surge frequency is matched iteratively, by varying the plenum volume size until the surge cycle frequency is matched.

NASA CC3 with vaneless diffuser - Transient reverse flow

Figure(5.66) shows the pressure ratio estimations of NASA CC3 compressor with vaneless diffuser during a deep surge cycle obtained by through-flow simulations at 100% speed. The steady-state, forward flow characteristic at design speed, derived from CFD simulations along with the reverse flow, CFD characteristics at 40%, 60%,

80% and 100% speed are included in the graph for reference. The red line indicates a qualitative, approximate trajectory of the surge cycle.

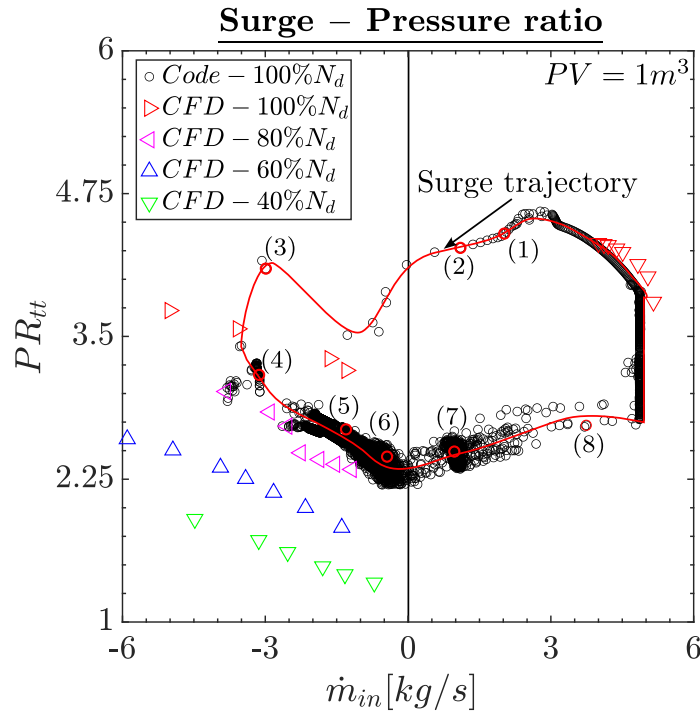


Figure 5.66: Pressure ratio versus mass flow during deep surge of NASA CC3 compressor with vaneless diffuser with a plenum volume of $1m^3$.

As the plenum pressurises, the mass flow decreases and the pressure in the flow is progressively augmenting, reaching the highest pressure rise at the point where the gradient of the characteristic becomes zero. The gradient then becomes positive and the pressure ratio only mildly decreases as the mass flow is approaching zero. Inherently non-axisymmetric instabilities such as rotating stall, typically manifest in the positive mass flow region, once the operating point moves beyond the surge line and the compressor stalls. This phenomena cannot be reproduced by the through-flow code due to the axisymmetric flow-field assumption, therefore the surge trajectory between peak pressure and zero mass flow is not adequately modelled. Despite these limitations, the continuous behaviour of the characteristics accompanied by mild pressure drops, is typical for centrifugal compressors (see section(1.4)), thus the actual surge cycle trajectory in that region is not expected to vary substantially. The pressure ratio traces follow a path which is more consistent with the 80% characteristic in the negative mass flow region of the map rather than the 100% one. This is due to changes in the corrected rotational speed, $N_d\sqrt{T_{ISA}/T_{0,in}}$, which at reverse flow conditions is corrected with the total temperature measured at the volute outlet. During reverse flow operation, the temperature in the plenum volume ranges between

[400K-480K] which corresponds to a corrected speed in the range [78% – 85%]. In practice, during the deep surge phase of the cycle ($\dot{m} < 0$), the operating point moves across different reverse flow characteristics due to changes in the plenum temperature and consequently the corrected speed. The physical time of the surge cycle is approximately 3.6 seconds corresponding to a approximately 1307 revolutions and a frequency of 0.28 Hz.

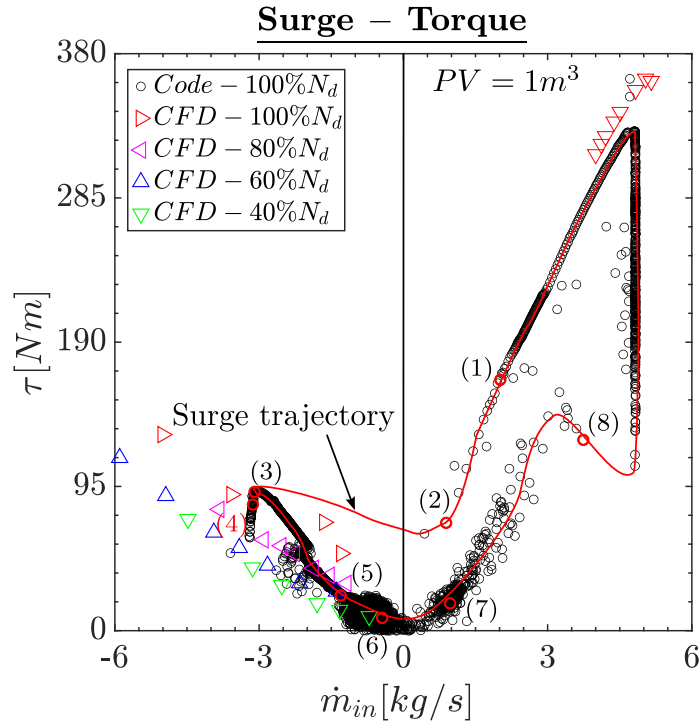


Figure 5.67: Torque versus mass flow during deep surge of NASA CC3 compressor with vaneless diffuser and a plenum volume of $1m^3$.

Figure(5.67) shows the torque traces along the same surge cycle. At forward flow conditions, the torque continuously decreases as the compressor moves towards stall since the torque is a function of the mass flow, the inlet and outlet radius and absolute circumferential velocity components (Euler turbine work). A gradual, linear reduction in torque is observed along the entire positive mass flow region. In the deep surge regime, the compressor operates at a lower corrected speed, approximately in the range [78% – 85%] which is also reflected on the torque reverse flow trajectory. The torque continuously decreases as the mass flow approaches zero and increases rapidly as the compressor recovers and the operating point shifts towards the higher positive mass flow regions. The torque traces are consistent with both the steady-state forward and reverse flow characteristics.

Streamwise velocity contours at different points during the surge cycle, annotated in figure(5.66) are shown in figure(5.68). Low-momentum flow, near the hub of the

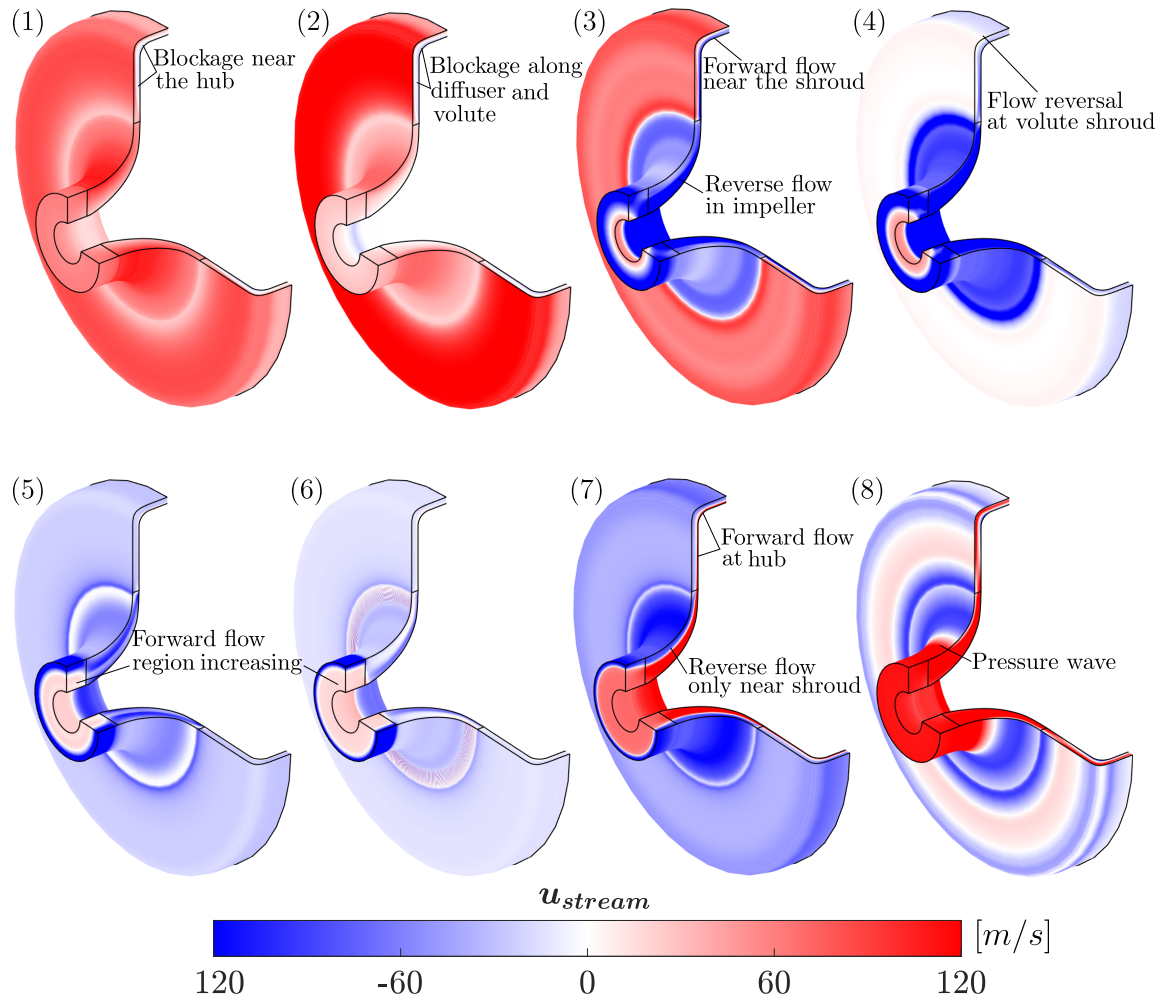


Figure 5.68: Streamwise velocity contours at different operating conditions during a deep surge cycle of NASA CC3 compressor with vaneless diffuser and a plenum volume of $1m^3$.

diffuser and volute regions (point (1)), develops initially in the positive gradient region. Further reduction in mass flow promotes the growth of the low-momentum region in the diffuser and volute region in both the streamwise and spanwise direction, generating substantial aerodynamic blockage (point (2)). The energy input into the flow is not sufficient to overcome the increased pressure downstream the impeller which results in flow reversal into the impeller (point (3)). A portion of the flow near the shroud, downstream the impeller, is still moving towards the outlet but is washed away a few instances later by a reverse flow stream near the physical outlet, at the volute shroud (point (4)). As the operating condition of the compressor moves down the reverse flow characteristics, the forward flow region at the physical inlet expands, suppressing the area near the shroud from which, the flow exits the blade passage (points (5) and (6)). Forward flow conditions gradually re-establish in the largest part of the blade passage, except for a persisting reverse flow stream near the shroud

which extends along the entire duct (point (7)). A pressure wave near the shroud removes the last reverse flow remnants and the compressor chokes shortly after.

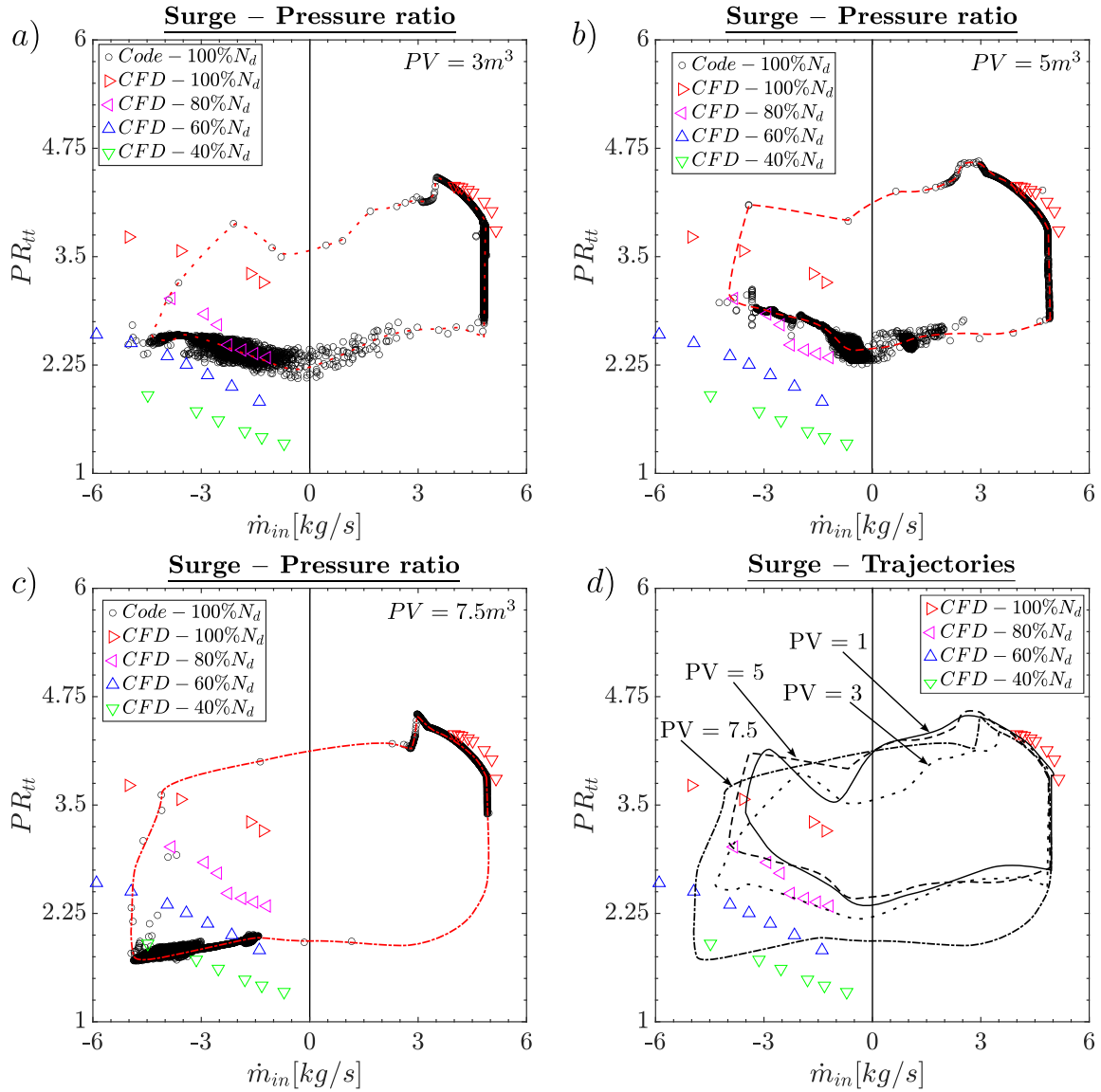


Figure 5.69: Pressure ratio versus mass flow during deep surge cycle of NASA CC3 compressor with vaneless diffuser and different plenum volumes: a) $3m^3$, b) $5m^3$ c) $7.5m^3$ d) comparison of surge trajectories.

The surge cycle behaviour is investigated by varying the size of the plenum. Three additional surge cycles are obtained by increasing the plenum volume to $3m^3$, $5m^3$ and $7.5m^3$ respectively. The pressure ratio traces of the 3 additional surge cycles, along with a qualitative comparison of the approximate cycle trajectories corresponding to all 4 plenum sizes are depicted in figure(5.69). The surge cycles corresponding to the three lower plenum volumes do not exhibit any major differences in the cycle trajectories. In the negative mass flow, high-pressure region (top-left on the map) very

few points were obtained due to the fast transient response of the compression system during flow reversal. The accuracy of the trajectories in that region, where the most noteworthy differences are expected due to the changes in the plenum volume size, are not captured with a sufficient number of points. During deep surge, the operating points move along a similar path, except for the higher plenum configuration, where the trajectory passes through the lower 40% and 60% characteristics, substantially increasing the trajectory of the cycle. Due to the large computational time of the largest plenum simulation, the obtained points introduce a degree of ambiguity to the definition of surge cycle path. It should be noted that the larger plenum volumes used here are very large with respect to the mass-flow capacity of the compressor as well as the volume of the downstream components e.g. combustor of an engine accommodating such a compressor. It is expected that during a deep surge event in an aero engine incorporating a centrifugal compressor of that size, the reverse flow path of the smallest plenum volume would be the most representative one.

NASA CC3 with vaned diffuser - Transient reverse flow

Following an identical approach, the transient, post-stall, aerodynamic behaviour of the NASA CC3 centrifugal compressor with vaned diffuser is investigated by performing through-flow simulations. The volume of the plenum is varied and deep surge cycles are obtained for 4 different plenum volumes: $1m^3$, $3m^3$, $5m^3$ and $7.5m^3$. The plenum volume is modelled following the 1D plenum-throttle approach, while open boundary conditions are imposed at the domain boundaries. The throttle is initially closed to pressurise the tank and shift the operating point towards the surge line. Once negative mass flow is detected, the throttle is fully opened.

Figure(5.70) shows the pressure ratio recordings during a deep surge cycle obtained with a plenum volume of $1m^3$. The forward flow characteristic at design speed and the reverse flow characteristics at 40%, 60%, 80% and 100% design speed are also plotted. In this configuration, the behaviour of the forward flow characteristic as the mass flow moves towards zero mass-flow changes to a near-zero gradient and does not exhibit the sudden pressure drop which the vaneless diffuser configuration exhibited at peak pressure ratio. The compressor goes into deep surge and follows a trajectory aligned with the 80% reverse flow characteristic. During flow reversal, the total temperature at the outlet ranges between 410K–490K and the corrected speed ranges between 0.77% – 0.83 %, which is in agreement with the CFD 80% reverse flow characteristic. The physical time of the surge cycle is approximately 3 seconds, corresponding to a approximately 1090 revolutions and a frequency of 0.33 Hz.

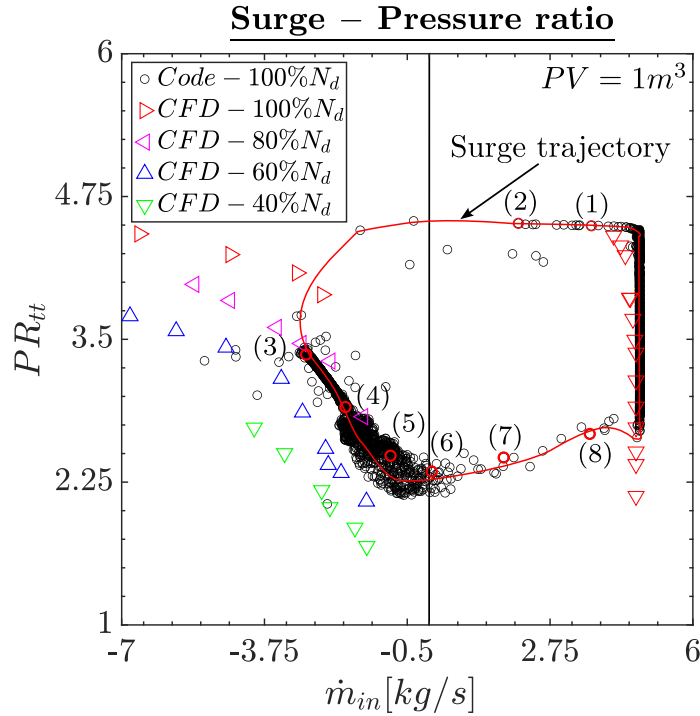


Figure 5.70: Pressure ratio versus mass flow during deep surge of NASA CC3 compressor with vaned diffuser and a plenum volume of 1 m³.

Figure(5.71) shows the recorded torque traces during the surge cycle. At forward flow conditions, the torque is in good agreement with the CFD results. Operating points at 80% and 100% torque characteristics are also included for reference although the convergence of the design speed reverse flow characteristics was not deemed adequate. In the positive, zero-gradient region of the pressure ratio characteristic, the torque drops abruptly and continues to decrease as it approaches zero mass-flow. In the highest negative mass flow region, the torque reaches its reverse flow peak value and keeps decreasing as it approaches zero mass flow. In the high-negative mass flow region, the torque is notably underestimated. This could be related to the torque extraction method which is based on the Euler turbine work and is therefore a function of the mass-flow which varies substantially at the inlet and the outlet during transients. An extraction method less sensitive to mass flow variations would be the extraction of torque directly from the body force distribution in the blade passage. Nevertheless, the shape and trajectory of the torque is consistent with the reverse flow torque characteristics which seem to collapse to a single characteristic at reverse flow conditions. The torque continuously increases once the operating setting of the compressor shifts towards the positive mass flow region of the map and increases rapidly once the compressor chokes.

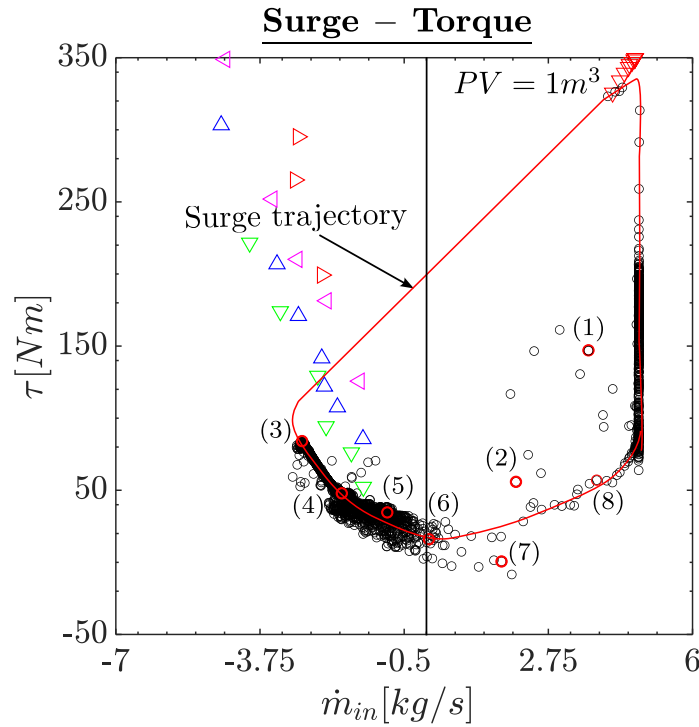


Figure 5.71: Torque versus mass flow during deep surge of NASA CC3 compressor with vaned diffuser with a plenum volume of $1m^3$.

The streamwise velocity contours at different instances during the surge cycle are illustrated in figure(5.72). The first point is captured in the zero-gradient region, showing a low-momentum flow in the volute near the hub region. Further increase in the static pressure of the plenum causes flow-reversal at the vaneless space and in an area in the vicinity of the vaned diffuser inlet and throat. This is in very good agreement with the experimental DPIV measurements by Wernet et al. [61, 62], who experimentally identified that flow reversal initiated at the exact same location using DPIV measurements (see figure(2.13)). In the low negative mass-flow region (point (3)), flow reversal gradually develops along the entire duct except for a small flow stream near the shroud and a part of the blade passage near the impeller TE. A few instances later, fully reverse flow conditions establish along the entire duct. As the plenum further de-pressurises, the forward flow region at the inlet increases, continuously suppressing the reverse flow area from which the flow exits the impeller (points (5),(6) and (7)). Forward flow conditions subsequently re-establish, preceded by notable streamwise velocity oscillations at the shroud and localised reverse flow remnants along the duct (point (8)).

The size of the plenum on the post-stall aerodynamic performance of the CC3 compressor with vaned diffuser is subsequently investigated by changing the size of the plenum to $3m^3$, $5m^3$ and $7.5m^3$. The raw pressure ratio data along with

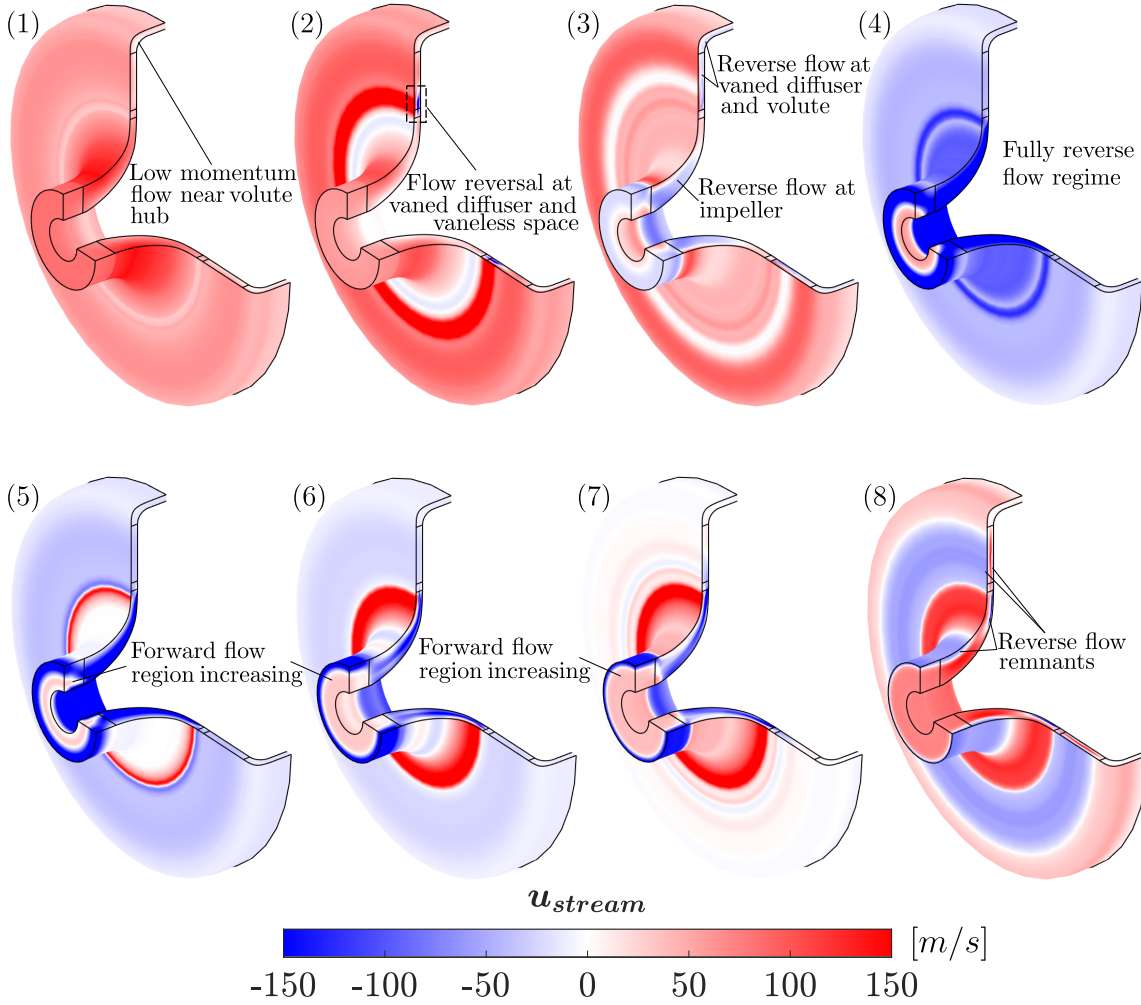


Figure 5.72: Streamwise velocity contours at different operating conditions during a deep surge cycle of NASA CC3 compressor with vaned diffuser with a plenum volume of 1m^3 .

a qualitative comparison of the approximate surge cycle trajectories are shown in figure (5.73). The surge cycles do not exhibit any noteworthy differences, with the reverse flow pressure ratio traces mainly being aligned with the 80% reverse flow characteristic. Due to the rapid transient behaviour from forward to reverse flow, not enough flow instances were recorded in the region between near-zero mass flow and the peak of the reverse flow characteristic, where a more extended surge cycle (starting higher up on the characteristic) is anticipated due to the increased plenum volume. Despite the limited number of obtained data, this effect is captured as shown in figure(5.73d). The obtained trajectories indicate that even at very large plenum volumes, the surge cycle is only passing through the lower-mass flow points of the reverse flow characteristics. During deep surge events, trajectories passing through the very high pressure regions of the steady-state reverse flow characteristics, obtained by steady-state reverse flow simulations (shown in the previous section) are deemed

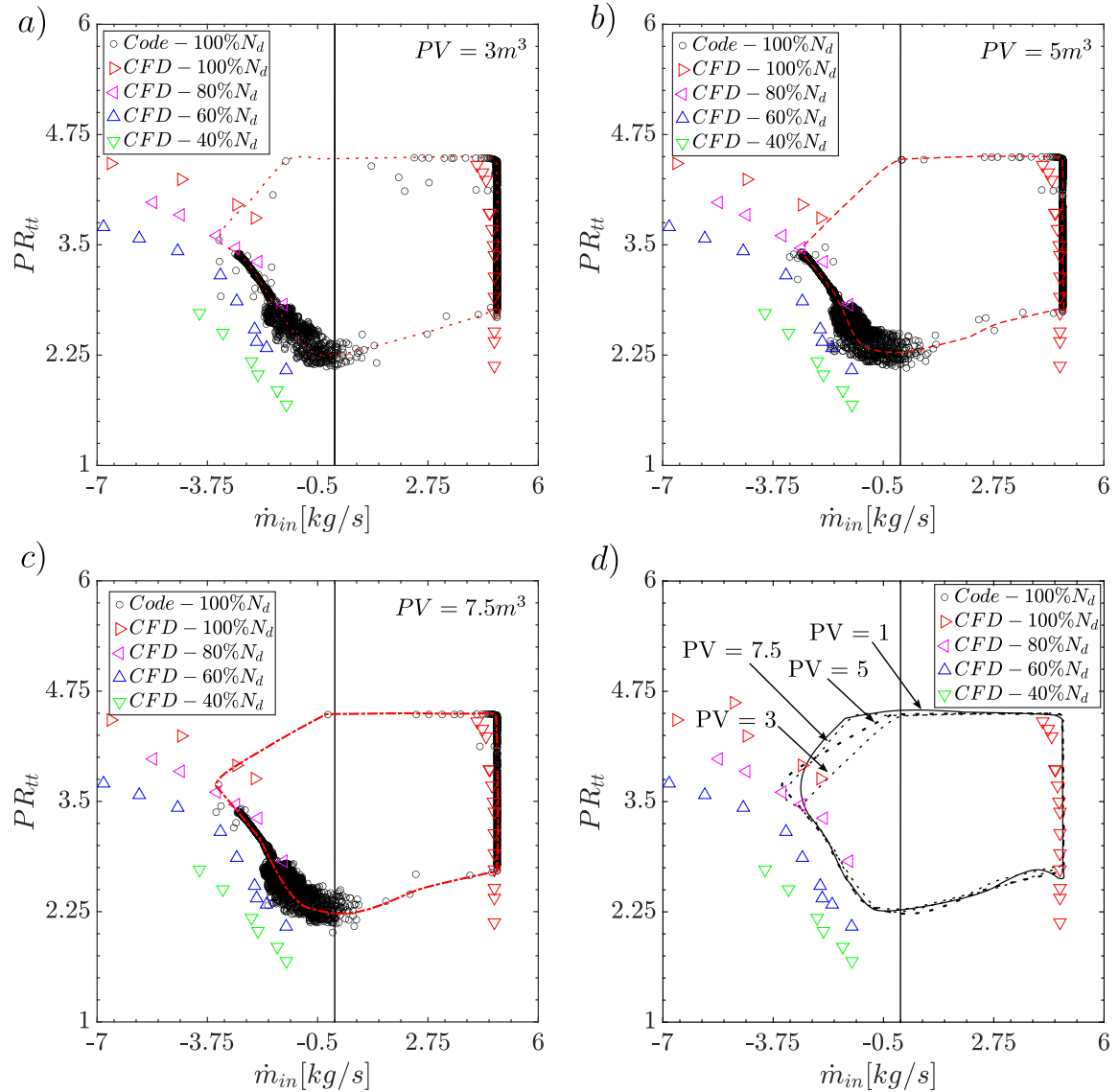


Figure 5.73: Pressure ratio versus mass flow during deep surge cycle of NASA CC3 compressor with vaned diffuser and different plenum volumes: a) $3m^3$, b) $5m^3$ c) $7.5m^3$ d) comparison of surge trajectories.

rather unrealistic, especially in an aero-engine accommodating compressor designs with centrifugal compressor stages, where the volume of the downstream components is smaller. Errors in the high-mass flow and high-pressure regions of the reverse flow characteristics are therefore not expected to notably affect the accuracy of post-stall transient predictions.

Experimental pressure traces obtained using fast-response dynamic pressure transducers of the NASA CC3 compressor during surge events are reported in [61, 62]. Data were experimentally obtained at three locations; vaneless space, vaned diffuser throat and impeller inlet [61, 62]. Pressure trace recordings were obtained for different

surge cycles. An extended inlet is attached in front of the impeller, while the presence of ducting and a plenum volume are reported. No geometric or sizing data regarding any of those components is provided. The reported frequency of the surge cycles is approximately 3Hz. The raw experimental pressure recordings at the vaned diffuser throat during a surge cycle event are shown in figure(5.74a). Figure(5.74b) shows pressure traces at the impeller inlet and the vaned diffuser during a different surge event with a lower frequency, indicating that flow from vaned diffuser travels back and exits from impeller inlet. Changes in the conditions under which the different surge events were obtained are not reported.

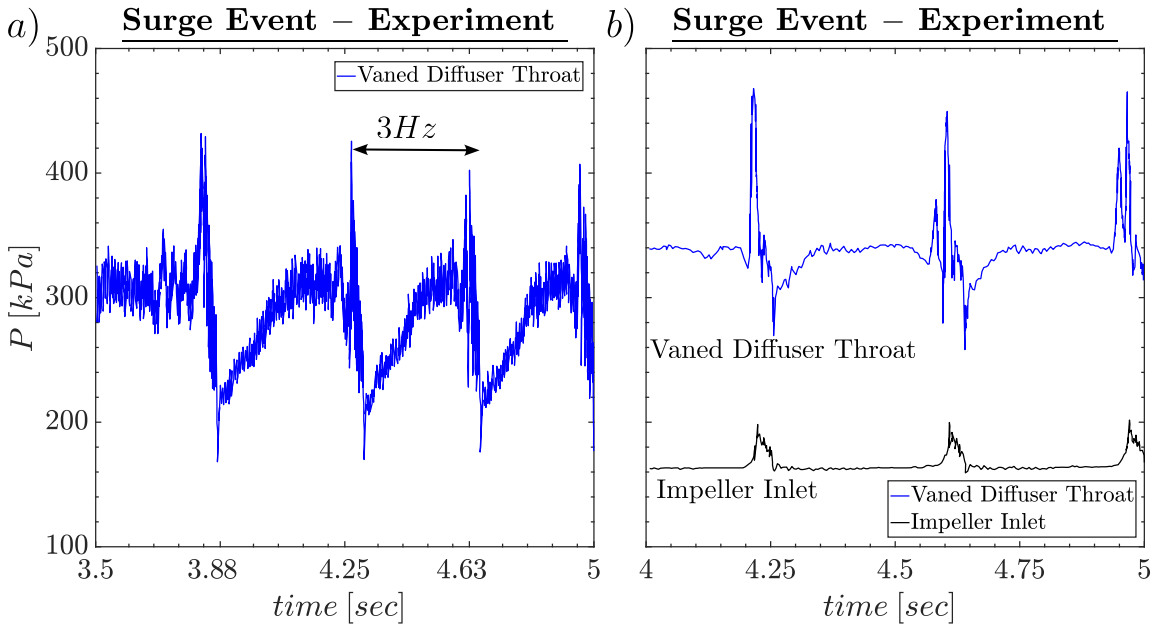


Figure 5.74: NASA CC3 experimental pressure recordings during two different surge events at a) vaned diffuser throat and b) vaned diffuser throat and impeller inlet (adapted from [61, 62]).

Due to the absence of data for the aforementioned geometrical components, no modifications to the through-flow computational domain are performed. Multiple surge events are simulated using the through-flow code coupled with an 1D plenum-throttle model, the volume of which, was varied until the frequency of the simulated surge cycles matched the experimental one. A plenum volume of $0.25m^3$ resulted in a cycle frequency of 3Hz, during which, 4 cycles are recorded. The throttle was 80% open throughout the entire surge event since the compressor would tend to recover after the completion of the first surge cycle when the throttle was fully open. Once the compressor stalled, the throttle was left open by 80% throughout the entire surge event. Figure(5.75) depicts the raw pressure recordings acquired by through-flow simulations at the diffuser throat and impeller inlet during two surge events. The

first event (left) corresponds to the compressor response with a plenum volume of $0.15m^3$ while the second (right) with a plenum volume of $0.25m^3$. The surge cycle frequency of the first event is 3 Hz while the frequency of the latter is approximately 4.5 Hz.

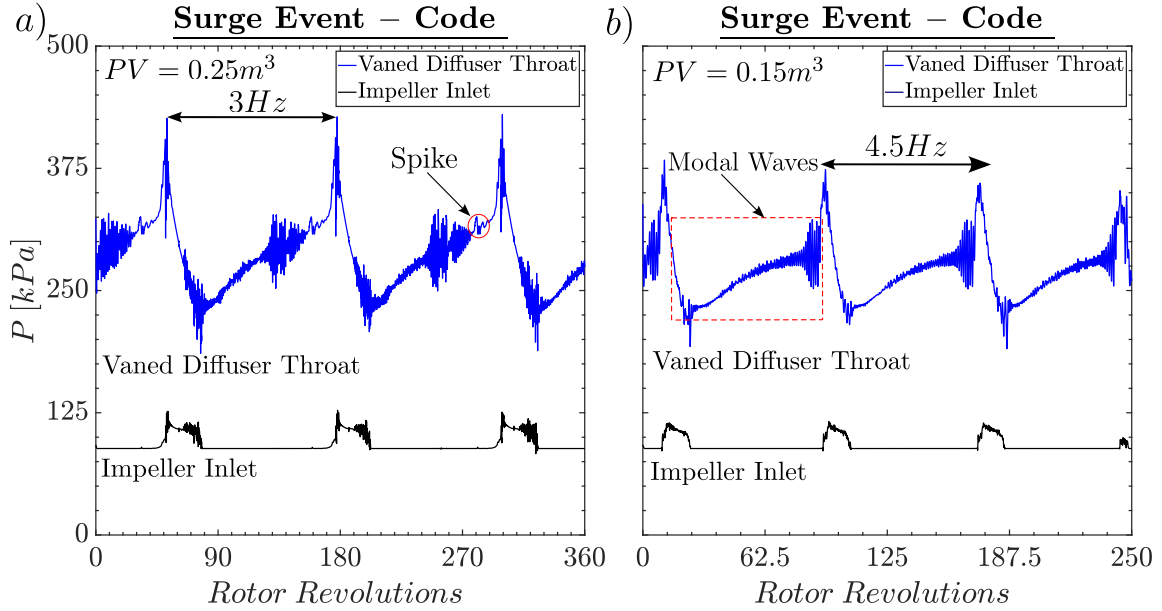


Figure 5.75: NASA CC3 through-flow pressure recordings at vaned diffuser throat and impeller inlet during surge events obtained with a plenum volume of a) $0.25m^3$ and b) $0.15m^3$.

The pressure traces at the inlet as well as the magnitude and the shape of the cycle pressure signals are consistent and in agreement with the experimental recordings. The experimental static pressure measurements at the impeller inlet, a few revolutions before the surge events, are lower than the simulated ones which could be related to differences in experimental conditions at the inlet of the impeller. The two surge cycles exhibit different patterns primarily in the region preceding the pressure peak. During the recovery phase of the first cycle, oscillations of increasing amplitude manifest. Their amplitude increases rapidly but the disturbances gradually dissipate. Shortly after the dissipation of the disturbances, a spike-type trace is shown, a few revolutions after which, the surge event initiates. This spike precedes all surge cycles of that particular surge event. Spike stall inception normally initiates from a single-passage instability which rapidly propagates in the circumference. The classification of that phenomenon as spike-stall inception could be deemed ambiguous, due to the axisymmetric nature of the solver. It is hypothesised that spike stall inception could be reproduced by the code, since spikes rapidly propagate around the circumference, growing into a local, axisymmetric disturbance shortly after their emergence in a

blade passage. Modal waves are primarily a circumferentially-independent type of instability thus can be captured by the code.

The second surge event (figure(5.75b)), shows a different stall inception pattern which resembles a modal wave-type of stall inception. Perturbations of increasing amplitude emerge during the re-pressurisation phase. These are not damped and grow rapidly a few revolutions before the initiation of the surge cycle. The stall inception mode is not reported in the experimental study [61, 62], thus no conclusive result can be made by inspection of the data to identify the validity of the stall inception mechanism predicted by the through-flow code.

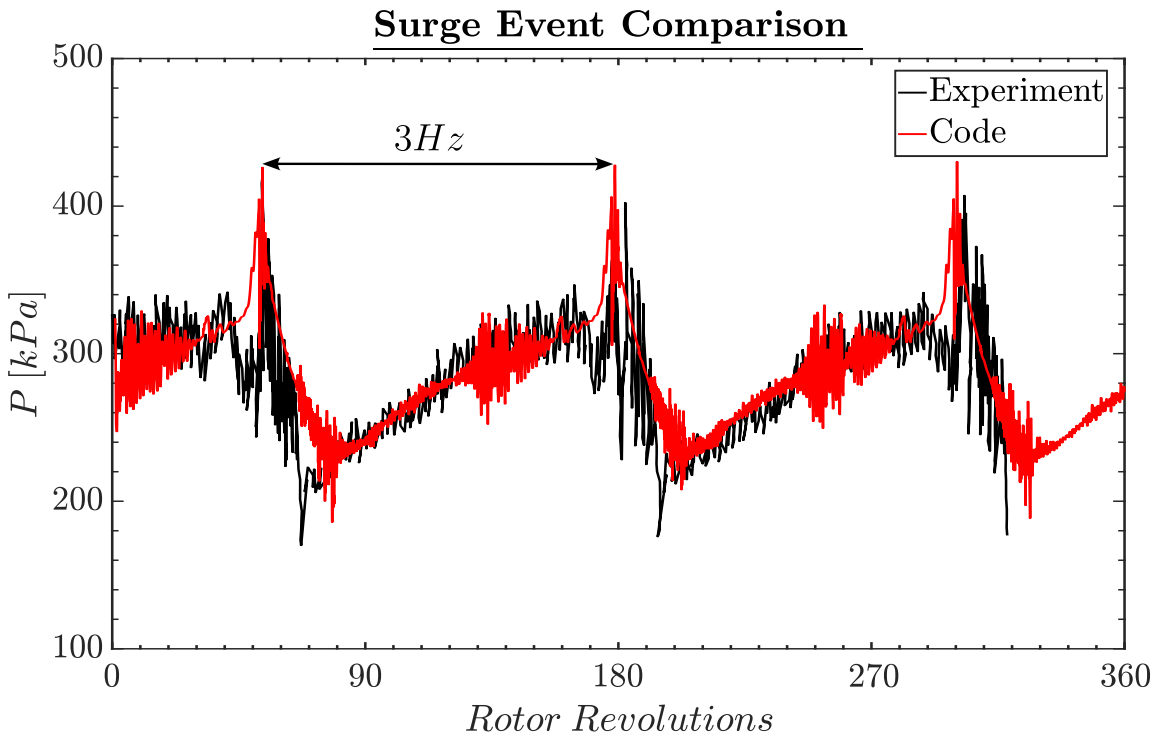


Figure 5.76: Comparison of NASA CC3 through-flow pressure recordings at vaned diffuser throat against experimental measurements during surge a surge event. Three consecutive cycles corresponding to the surge event are shown.

To assess the validity of the simulated 3Hz surge cycle, the recorded pressure traces are compared against the experimental pressure recordings at the vaned diffuser inlet corresponding to the 3Hz experimental surge event. Figure(5.76) shows a comparison between the experimental static pressure measurements at the throat of the vaned diffuser and the corresponding recordings obtained from body force simulations. The experimental data correspond to the last 3 surge cycles depicted in figure(5.74a). The simulation of the entire surge event was completed in less than 6 hours, running on a single core of a regular desktop computer's processor. The simulated pressure traces are in excellent agreement with the experimental results. The magnitude of

the higher and lower pressure peaks are very accurately predicted by the through-flow code. The most noteworthy discrepancy between the experimental and simulated cycles is the duration of the region between the highest and lowest pressure peaks, the duration of which, is slightly overestimated by the code. The simulated instability which emerges and subsequently dissipates during the recovery phase is not evident in the experimental results but is not affecting the dynamic behaviour of the compressor. As discussed in literature review section, the dynamic response of the compressor is dependent not only on the volume but also on the geometry of the components. It is hypothesized that the inclusion of the experimental inlet duct could potentially result in improved accuracy. The highly oscillatory behaviour of the pressure during certain phases of the cycle is hypothesised to be partially related to insufficient mixing, causing an unnaturally prolonged presence of a reverse flow stream near the shroud, especially during the recovery phase of the cycle. For reference, the pressure ratio and torque measurements during the simulated surge events are shown in figure(5.77), indicating a deep surge event. The surge cycle confirms the researchers' conjectures about flow from the diffuser travelling all the way back to the impeller inlet.

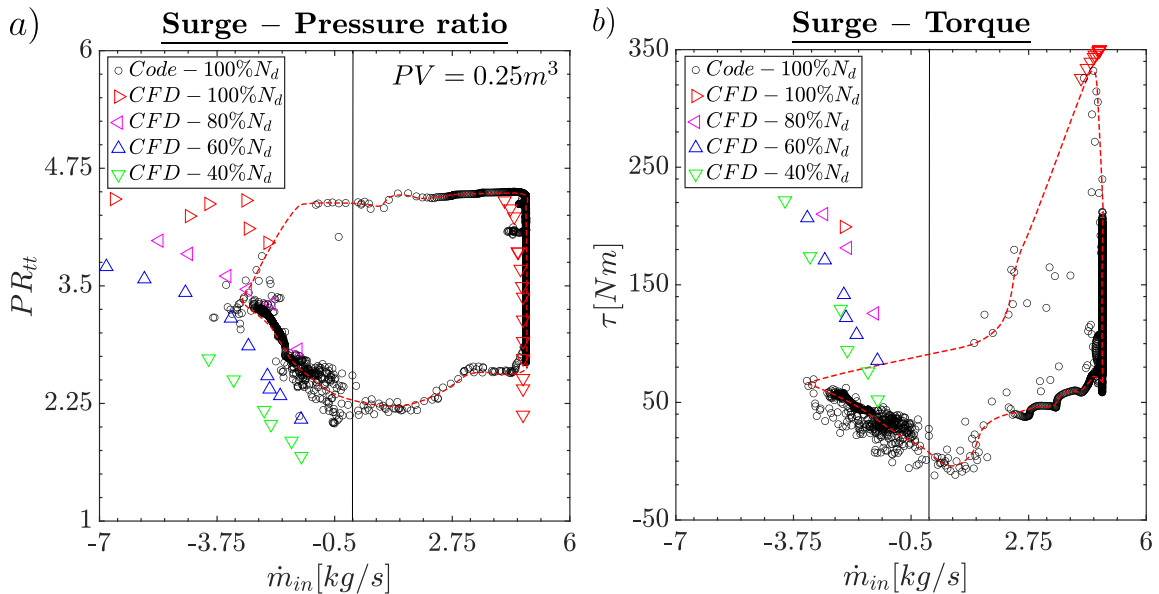


Figure 5.77: a) Pressure ratio and b) torque versus mass flow of the simulated experimental surge event of NASA CC3 compressor with vaned diffuser and a plenum volume of $0.25m^3$.

Pressure traces (presumably filtered) at the same location during another surge event are shown in figure(5.78). Details regarding the frequency of the surge cycle are not reported but both the values of the highest and lowest peaks are lower than the ones shown in figure(5.76) indicating a slightly lower frequency cycle. Pressure profiles during a surge cycle obtained with a plenum volume of $0.2m^3$ and $0.25m^3$

are compared against experimental results. For clarity, the pressure traces are filtered. Both profiles correlate exceptionally with the experimental traces. The profile corresponding to the smaller plenum volume, more precisely approximates the experimental traces, accurately estimating both the magnitude of the pressure peaks and the duration between them. The large oscillations preceding the cycle are not evident in the experimental results although they could have been filtered out. The profile corresponding to the higher plenum volume is wider, while in both cases, the pressure during the recovery phase is slightly overpredicted.

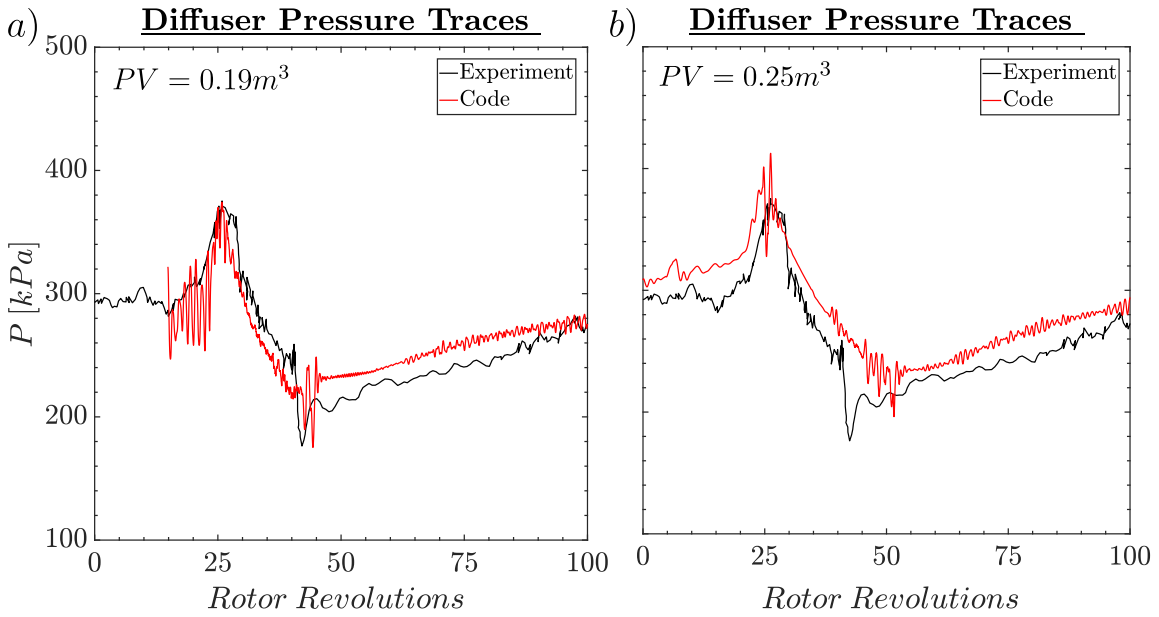


Figure 5.78: Comparison of experimental and simulated pressure traces at vaned diffuser throat during a surge cycle with plenum volume of a) 0.19m^3 and b) 0.25m^3 .

5.3 Chapter summary

In this chapter, the validity and limitations of the overall through-flow modelling approach are discussed. The pre and post-stall, steady-state and transient accuracy of the through-flow method is assessed by extracting global compressor performance metrics, spanwise profiles and flow-field contours from body force simulations and comparing them against CFD and experimental data. Forward flow validation is performed against 5 compressor geometries: NASA rotor 37 and stage 67 (axial rotor and fan stage), DLR SRV2-0 (centrifugal compressor with vaneless diffuser), CSTAR (centrifugal compressor with vaned diffuser and volute) and NASA CC3 (centrifugal compressor with vaneless and vaned diffuser and volute). The results reveal a high dependence of the code's global aerodynamic performance estimations on the accuracy

of the axial compressor correlations, which in turn, are highly sensitive to the tuning coefficients. Without tuning, the pressure ratio characteristics of both axial geometries are predicted reasonably well, but the efficiencies are significantly underpredicted with the corresponding errors reaching 12% and 20% for rotor 37 and stage 67 respectively. Tuning of those parameters provides substantial accuracy benefits, reducing the efficiency error of stage 67 to less than 3.1%. The performance predictions of all modelled centrifugal compressors is obtained without prior tuning of the loss and slip models and are in very good agreement with the experimental and CFD results. High discrepancies with respect to the SRV2-0 experimental data are observed for both the through-flow and the CFD predictions, mostly at higher rotational speeds. Both the choking mass flow and the pressure ratio are substantially overpredicted, but this is consistent with the observations of several authors in the literature. Metal and aerodynamic blockage is an important modelling aspect in centrifugal compressor modelling for accurate choking mass flow predictions. The shape of the characteristics at lower speeds tends to be consistently flatter, not capturing the steeper gradient near the high mass-flow region. This behaviour is observed, although significantly more pronounced, in body force-based centrifugal compressor simulations in literature [49, 85]. The spanwise flow-field distributions in axial compressor passages are more sensitive in operating condition changes compared to impeller and vaned diffuser blades. The inclusion of a single distribution coefficient for all operating points introduces notable errors in axial compressor blades but is a valid modelling approach for impeller and vaned diffusers. The limitations of the coefficients are predominantly related to the underestimation of losses near the hub and the tip which in turn, are primarily related to limitations of the viscous force extraction method. In the volute duct, downstream the diffuser, significant mixing takes place, transforming the highly non-uniform profile at the TE, to an almost uniform one. In the absence of explicit mixing modelling and long ducting components, a simple uniform coefficient is able of accurately reproducing that effect. The flow-field is captured with reasonable accuracy, except for localised, viscous-related phenomena such as separation regions near the shroud. Steady-state reverse flow characteristics of SRV2-0 and the two NASA CC3 configurations are obtained with the code and compared against CFD data. The SRV2-0 pressure and torque characteristics obtained by the code are consistently underpredicted which is consistent with the forward flow results. The CC3 characteristics of the vaneless diffuser configuration are in very good agreement with the CFD results in the lower mass flow region and less accurate in the higher one, due to overprediction of the aerodynamic blockage. Excellent agreement between the

2 models is achieved in the configuration with the vaned diffuser. The visualization of the flow-field indicates the presence of a large recirculation region in the inlet duct and a smaller one at the hub, near the TE along with the formation of shock waves at the TE, in the higher spanwise positions. The through-flow representation of the flow-field correlates reasonably well with the CFD one, capturing all the essential flow-features, with the most notable deficiency being the non-uniformity of the flow inside the impeller passage, due insufficient mixing and the swirl of the flow inside the blade passage which cannot be captured by the code. Deep surge cycles are obtained with both NASA CC3 configurations by varying the plenum volume size. The trajectory of the cycles are consistent with the steady-state, CFD reverse flow characteristics, while the corrected rotational speed decreases during reverse flow operation due to changes in the plenum temperature. Spike and modal stall inception is captured when different plenum volume and throttle settings are imposed. A surge cycle of the CC3 compressor with vaned diffuser is simulated by varying the plenum volume until the experimental surge cycle frequency is matched. The shape, frequency and magnitude of the experimentally recorded pressure traces at the diffuser throat are matched exceptionally well. The pressure traces at the impeller inlet are also successfully captured, while flow-field contours show the emergence of flow reversal in the vaneless space and the diffuser throat which is in agreement with the experimental DPIV measurements. The simulation of the entire surge event required approximately 6 hours on a regular computer, running on one core. Simulation of the same event using conventional CFD methods would be expected to require multiple days and even weeks on multi-core HPC systems.

Chapter 6

Conclusions and future work

Contents

6.1 Summary	325
6.2 Conclusions	328
6.3 Contribution to knowledge	332
6.4 Recommendations for future work	334

6.1 Summary

The aim of the present work is the development of a lower-order, body force-based approach to predict the pre and post-stall aerodynamic performance of axial and centrifugal compressors while drastically reducing the associated numerical cost. The project primarily focuses on centrifugal compressor modelling, due to the substantial research gap regarding through-flow modelling methods in centrifugal compressors. A successful and experimentally validated, rotating stall and surge through-flow modelling approach for multi-stage axial compressors was recently developed in Cranfield university [27, 30, 34, 36, 58, 92, 93], forming the foundations of the current project.

The governing equations are modified, encompassing a coalescence of unique features. Conceptual modifications are introduced in the inviscid, axisymmetric Euler equations; a blockage parameter is introduced to account for decrements in the local effective area due to blade and aerodynamic blockage, simplified shear stress terms to enable flow mixing and body force source terms to introduce flow turning and losses. Metal and aerodynamic blockage modelling was deemed essential due to the large thickness of impeller and vaned diffuser blades and due to the

substantial aerodynamic blockage, which in centrifugal compressors is large even at forward flow conditions. The governing equations are solved using Godunov's scheme and 3 Riemann solvers in the rotating FoR, providing significant numerical stability benefits and shock capturing capabilities. The grid is constructed via the synthesis of subdomains, the resolution of which, is adjusted by changing the number of streamwise elements in individual subdomains. The mesh generation algorithm ensures the construction of grids aligned with the meridional projection of the blade's LE and TE preventing locally discontinuities in the body-force fields. An explicit body force model was derived on the basis of fundamental, streamline curvature principals, following primarily the ideas of semi-explicit methods developed at MIT [85, 86, 88]. The new model is applicable to all compressor blades and is capable of transiently imposing losses as well as introducing flow turning at forward, reverse and mixed flow conditions with exceptional precision. Distribution coefficients are introduced and mathematically defined to more accurately distribute deviation angles and losses along the blade passage. Distributions extracted from CFD simulations are used exclusively or in conjunction with simple, custom ones. Depending on the modelled compressor blade component, dedicated loss and flow deviation/slippage predictions are transiently obtained at forward flow conditions by empirical correlations. Analytical models for the estimation of aerodynamic blockage as well as losses and flow angle changes at reverse flow conditions due to mixing-out of the jet and separated flow are derived for both axially and radially positioned blades. An overall improved reverse and mixed flow modelling approach is implemented and is applicable to all types of compressor blades.

Extensive investigation of the validity and limitations of the individual modelling aspects as well as of the entire through-flow code is carried out in the results section. The verification of the solver is initially presented. A validation and comparison of the implemented Riemann solvers is performed against the analytical solution of Sod shock tube-type of problems. The flow-field solution of an empty, centrifugal compressor duct is obtained by the developed axisymmetric Euler solver and compared against the flow-field solution obtained by ANSYS FLUENT, using an identical grid. The correct implementation of the modified governing equations with blockage in both the absolute and rotating FoR is subsequently investigated. The numerical implications introduced due to blockage modelling are discussed and addressed using shape functions, preventing abrupt changes in the cell faces areas and volumes. The capabilities of the turning force model are demonstrated on an axial fan rotor and stator blade, an impeller and a vaned diffuser blade. The implications of over-

stiffening the flow with high correction forces is investigated. Verification of the viscous force is performed on an impeller and an axial rotor by imposing a range of prescribed pressure losses and comparing them against the resulting, mass-flow averaged ones. The distribution coefficient method is verified by imposing a range of prescribed deviation angles and pressure losses and distributing them in the blade passage using combinations of custom and CFD-derived coefficients. The extracted, mass-flow averaged deviation angles and pressure losses are compared against the prescribed ones. Pressure and efficiency characteristics at 60% speed are also obtained using different combinations of CFD and custom coefficients to assess the impact of distribution coefficients in global aerodynamic performance predictions. Subsequently, the validity and limitations of the new blockage-mixing and loss model are evaluated. The losses along the blade passage during reverse flow operation are extracted from steady-state CFD simulations and compared against the losses introduced in the through-flow code. The losses downstream the LE due to incomplete mixing-out of the flow are compared against CFD results but the presence of large recirculation regions at the inlet domain impeded the acquisition of reliable measurements. The aerodynamic blockage predictions of the new model are compared against 2 other models reported in the literature [33, 91] using data from CFD flow-field solutions. The flow deviation during reverse flow conditions is extracted along the blade passage to assess the validity of the zero-deviation angle assumption of the blockage-mixing model. The impact of the losses, modified curvature and deviation on the position and shape of the pressure ratio and torque reverse flow characteristics is analysed.

Forward flow validation of the through-flow code is performed by comparing global compressor aerodynamic performance quantities against CFD and experimental data on five compressor geometries: an axial rotor, an axial fan stage and three centrifugal compressor geometries. The accuracy of the flow-field solution is further assessed by comparing spanwise pressure ratio, temperature ratio and efficiency profiles against CFD ones, as well as via a qualitative comparison of different flow-field variable contours. Steady-state, reverse flow characteristics are obtained for three centrifugal compressors and are compared against CFD data. The ability of the code to replicate essential reverse flow features is assessed by examining the flow-field extracted from through-flow and CFD simulations. The post-stall transient response of the two NASA CC3 configurations (vaned and vaneless diffuser) is finally simulated using an 1D plenum-throttle model. Deep surge cycles are acquired by varying the volume of the plenum and the surge cycle trajectory is assessed on the basis of the forward and reverse flow characteristics. Surge cycles of the NASA CC3 compressor with

vaned diffuser are simulated with the through-flow code. Stall inception due to both spikes and modal waves are reproduced by the code when the volume of the plenum is varied. The code is validated against experimental pressure traces, recorded during surge events of the NASA CC3 compressor, matching exceptionally well both the frequency and the pressure measurements. The through-flow code enables the simulation of complex surge events at a fraction of the time that conventional CFD modelling methods would require. The computational cost/time benefits are the same in forward and reverse simulations and are typically at least an order of magnitude lower compared to conventional CFD methods.

6.2 Conclusions

The most essential conclusions of the present work are summarised below:

- The implementation and comparison of approximate and exact Riemann solvers designates that the additional computational cost and robustness downsides associated with exact Riemann solvers, outweigh the accuracy benefits they provide. More sophisticated Riemann solvers e.g. HLLC, Roe etc. provide noteworthy accuracy benefits compared to simpler ones e.g. HLL, while being computationally efficient and robust. Compared to exact Riemann solvers the accuracy of approximate ones is marginally worse. The HLLC method was found to be computationally more efficient than the Roe-Pike approach. Shock waves are very accurately captured by the developed solver.
- The additional blockage gradient terms arising in the governing equations with blockage, are evaluated with exceptional accuracy by the new implicit method, overcoming the limitations of finite difference methods reported in the literature [38]. Numerically-induced pressure losses of up to 4.8% related to blockage modelling manifest due to abrupt changes in cell face areas and volumes at interfaces adjoining bladed and ducted domains (with and without metal blockage). This is more pronounced in impellers and is further exacerbated in vaned diffuser blades due to their large thickness and the substantial change in radius which introduces large variations in the cell face areas. This is addressed by gradually increasing the blockage near the LE and TE, reducing the pressure losses to less than 1.8%.
- The body force model is extensively validated against different compressor blade geometries including axial fan rotors, stators, impellers and vaned diffuser

blades. The misalignment of the flow with respect to the imposed relative flow-path is trivial and does not exceed 1% at the TE in all tested geometries. The degree of interference of the correction force is crucial since it might over-stiffen the flow causing unrealistic behaviour with large under- and overshootings. Errors in the imposed pressure losses associated with the viscous force model are increasing as the imposed losses increase. Within the lower pressure loss region, representative of the losses in axial and impeller blade passages, the recorded errors are within a [0% – 5%] range.

- The distribution coefficient concept is able of capturing inherently viscous-related phenomena such as the mixing out of the tip leakage vortex and boundary layers and recasting them into the through-flow solution. By applying a prescribed range of flow deviation angles and pressure losses and applying different distribution coefficients, it is demonstrated that despite the drastic changes in the spanwise pressure ratio and temperature ratio profiles, the mass-flow averaged deviation and pressure losses are matching the imposed ones, with the corresponding errors not exceeding 2.6% and 4% respectively. This is further verified by comparing pressure and efficiency characteristics using different coefficients. The distribution coefficients rearrange the spanwise distribution profiles of the quantities of interest but do affect the averaged flow-field and therefore global aerodynamic performance.
- Forward flow performance predictions are carried out in axial and centrifugal compressor geometries. Accurate performance predictions of axial blade passages is shown to be highly dependent on the accuracy of the empirical correlations. Tuning of the correlations resulted in a reduction of the peak efficiency errors from 20% to 2.5% and pressure ratio errors lower than 3.1% for NASA stage 67. The predictions of the tested centrifugal compressor configurations are in good agreement with the CFD and experimental results, especially in the lower NRT range. Substantial overprediction in mass flow and pressure ratio of the SRV2-0 compressor is predicted by both the CFD and the through-flow code at design speed but these discrepancies are also reported in the literature. In centrifugal compressors with vaned diffusers, aerodynamic blockage is found to be essential for the prediction of the choking mass flow. At low speeds, the gradient of the characteristics near the highest mass flow region tends to be less accurate, presumably due to underestimation of aerodynamic blockage and upstream influence induced by the blades. This was also reported in the literature. On

average the errors are not exceeding 5% but errors up to 7% in choking mass flow 9% in pressure ratio and 7.5% in efficiencies are reported (higher SRV2 design speed).

- The forward flow spanwise distributions are in good agreement with the CFD ones. The inclusion of CFD-derived coefficients is a decisive aspect for the accurate reproduction of spanwise flow-field variations. The profiles of axial blade passages do not vary substantially between choking and off-design conditions but change dramatically near stall operation. This is not the case in impeller blades, where the changes in the spanwise profiles along the entire characteristic are rather insignificant. This indicates that a single distribution coefficient would provide a rather accurate representation of the entire operating range in impeller and vaned diffuser blades but not in axial blades operating at near stall conditions. The losses near the hub and the shroud are consistently underestimated by the through-flow code in all tested geometries which is reflected on the higher local errors and higher efficiencies. This is related to the assumptions of the viscous force extraction approach and the way it is incorporated into the viscous force model. The importance of boundary layer growth and flow deceleration in the diffuser is also highlighted; in the diffuser the profiles become more concave, losses increase near the hub and the shroud and the profiles become more uniform due to mixing effects. This is particularly evident in the volute where the mixing of the flow results in almost uniform spanwise profiles. In the absence of mixing modelling (inviscid flow), whenever long ducted domains downstream the last blade row are present, the inclusion of a simple uniform distribution at the last blade passage is essential to accurately reproduce the mixed-out, uniform profiles, whenever
- Aerodynamic blockage predictions at reverse flow conditions are obtained with the new model as well as other two models reported in the literature using CFD flow-field solutions. The predictions of the new model are consistent with Longley's, while the behaviour of the Powers et al. model is primarily dictated by the stall parameter and tends to predict excessive blockage values at very high incidence angles. The deviation at the LE during reverse flow operation is typically not exceeding 5° and diminishes at higher PR settings. Higher deviation angles are associated with downstream phenomena e.g. recirculation regions and vortical structures, generating blockage. Impellers are more prone to high deviation angles at the LE due to the absence of splitter blades near

the LE. The average pressure losses along the blade passage are expressed via rotary stagnation pressure losses to remove mean-streamline radius effects. The distributions of the pressure losses along the blade passage are in very good agreement with the CFD ones, but higher losses are generally introduced by the loss model. Due to the high non-uniformity of the flow near the LE, the validity of the blockage-mixing model is not properly assessed but promising results are obtained from preliminary comparisons.

- The isolated and combined impact of aerodynamic blockage, deviation and losses on the reverse flow characteristics is investigated. Aerodynamic blockage deviation and losses introduced by the turning force are the dominant quantities, dictating the mass flow capacity and pressure ratio. The modified curvature determines the extent of the misalignment region, substantially altering the choking mass flow and the losses. Higher losses by up to 10% are reported when the modified curvature region is extended from 10% to 60%. The impact of additional mixing losses are of second order compared to flow deviation and the losses in the passage.
- Reverse flow characteristics are obtained for 3 centrifugal compressor configurations via through-flow and CFD simulations. The recorded torque and pressure ratio discrepancies between the through flow and the CFD SRV2-0 compressor models are in the [2% – 10%] range, while the largest errors are recorded at higher rotational speeds. The reverse flow characteristics of the NASA CC3 compressor with vaneless diffuser indicate an overestimation of the aerodynamic blockage along the entire operating range but the results are in very good agreement in the lower mass-flow regions. In the configuration with the vaned diffuser, the characteristics match exceptionally well, especially in the lower mass flow region. The flow-field contours indicate the presence of a large recirculation region at the inlet domain and a smaller one near the impeller TE, the size of which, is governed by the pressure gradient along the duct. Blade to blade flow-field contours show the existence of separated flow regions and shock waves at the impeller TE near the shroud, along with vortical structures convecting back to the vaned diffuser inlet. The through-flow contours confirm that the reduced-order model is capable of successfully reproducing essential flow-features.
- Surge cycles with different plenum volumes are performed for both NASA CC3 configurations at design speed. During flow reversal, the temperature at the

physical outlet varies, with the time-averaged temperature being substantially higher than the atmospheric one. The compressor operates at a lower corrected speed, which is successfully captured by the surge cycle trajectory which is consistent with the CFD pressure ratio and torque characteristics. Experimental pressure traces at the vaned diffuser throat during surge events of the NASA CC3 compressor constitute the validation test case used for surge validation. Since the plenum volume is not provided, a plenum volume resulting in similar surge frequency is iteratively identified. The pressure traces at the vaned diffuser are in very good agreement with the experimental data, successfully capturing both the magnitude and the dynamics of the event. Streamwise velocity contours indicate that the code accurately predicts the inception of flow reversal in the vaned diffuser and vaneless space matching the experimental DPIV measurements. Modal and spike stall inception is simulated by the code, when plenum volume and throttle settings are varied.

- The through-flow modelling approach provides a drastic reduction in computational cost, enabling the simulation of surge events within a few hours. The computational cost is at least an order of magnitude lower compared to regular CFD methods and can be further improved by parallelising the code and amending the structure of the code.

6.3 Contribution to knowledge

A lower-order, body force-based, modelling approach is developed, aiming to predict the pre and post-stall aerodynamic behaviour of aero-engine axial and centrifugal compressors. The accomplishment of the project's aims and objectives, necessitated the expansion and advancement of methods reported in the literature along with the development of new ones. The different modelling approaches are unified in the form of a through-flow code and its capability of accurately predicting compressor aerodynamic performance at forward and reverse flow conditions is demonstrated. The evident research gap regarding lower-order modelling methods in centrifugal compressors, shifted the research primarily towards centrifugal compressor modelling. Nevertheless, the developed methods are applicable to both axial and centrifugal compressor types of blades. Key novelty aspects of the projected are:

- The governing equations are solved in the relative FoR and are conceptually modified to consider blockage and mixing effects. The governing equations with

blockage are derived, while the fluxes are obtained using the Godunov scheme coupled with 3 Riemann solvers, which are reformulated to ensure compatibility with the rotating FoR. The modified governing equations encompass a unique amalgamation of characteristics which is not reported in any relevant study in the literature.

- A new method for the estimation of the additional blockage terms is proposed. It enables the evaluation of the terms in ducts with large gradient changes with exceptional accuracy, thereby overcoming the limitations of finite difference methods reported in the literature.
- A new, blockage-mixing model at reverse flow conditions is developed. The model estimates the aerodynamic blockage due to flow separation along with losses and changes in the flow direction downstream the LE due to mixing phenomena and is applicable to both axially and radially positioned blades. Aerodynamic blockage effects are introduced into the flow-field transiently by varying the blockage parameter accordingly.
- A new body-force model is developed, comprising a base turning force, correction force and a viscous force component. The turning force model accounts for the full curvature of the blades in 3D, including blade lean effects and is applicable to all types of blades. The viscous force model accounts for the flow-path curvature of the duct and is applicable to axial and centrifugal compressor blades. The incorporation of loss correlations and the optional incorporation of custom distributions makes it a fully explicit model, completely independent of other inputs.
- A new approach for the mathematical definition and extraction of distribution coefficients for certain quantities is proposed. The distribution coefficients are coupled with the body force model providing realistic distributions of flow deviation and losses along the blade. These provide a substantial improvement over the simple, one-directional coefficients reported in the literature which are based on sinusoidal or linear distributions. CFD coefficients are capable of recreating complex, viscous phenomena, such as mixing-out of tip leakage vortices and boundary layers, significantly improving the accuracy of the simulated flow-field.
- An improved, mixed and reverse flow modelling scheme is developed on the basis of axial blade schemes. This adjoins aerodynamic blockage, additional losses

and flow deviation due to mixing-out of the flow as well as blade passage losses which are introduced via the turning force. The latter has been modified by adding modified curvature regions at mixed inwards and reverse flow conditions providing more control over the extent of the separation region and therefore the mass flow capacity and imparted losses. This constitutes the first, forward-mixed-reverse flow modelling approach applicable to all types of blades.

- The code constitutes the first successful through-flow code capable of predicting the pre and post-stall aerodynamic performance of centrifugal compressors. Only two partially successful attempts are reported in the literature. The first relies exclusively on CFD data for the extraction of all forces and only a few points at a low speed characteristic were obtained, while convergence issues were encountered at higher speeds. Accuracy near choke and stall was poor. The second, was successfully applied to a radial impeller using a semi-explicit approach, thus is dependent on extensive CFD data. The developed code is able to predict the pre and post-stall, steady-state and transient behaviour of centrifugal compressors without relying on CFD input (apart from distribution coefficients which are optional) and without being subject to the limitations of the aforementioned methods. The proposed modelling approach addresses all the limitations reported in the literature and constitutes the first successful and validated through-flow method capable of predicting the pre and post-stall aerodynamic performance of centrifugal compressors.

6.4 Recommendations for future work

Recommendations for further research, relevant to the present thesis are listed below:

- Improvements in the grid generation algorithm to enable the spanwise gridlines to adapt to leading and trailing edges with substantial curvature on the meridional plane. The linear assumption is valid for impeller, vaned diffuser blades and smaller axial compressor blades but modern fan blades are characterised by substantial curvature at the LE and TE in the meridional plane. Further grid improvements, using a more modular grid generation approach, could enable the construction of grids even when other geometries are intervening the duct e.g. core and bypass duct.
- The conversion of the solver into a fully-3D one, capable of considering circumferential gradients. This is a prerequisite numerical aspect for rotating stall

modelling since it is an inherently non-axisymmetric phenomenon and cannot be captured by the current solver.

- The Godunov scheme currently used in the code is first order accurate in space and time, thereby introducing substantial numerical diffusivity. The second order variation of the scheme (MUSCL) is second order-accurate, thereby reducing the associated numerical diffusivity. Unlike first-order schemes which inherently are monotone, higher-order schemes are coupled with flux limiters to prevent numerical instabilities and spurious oscillations.
- To trigger rotating stall, an algorithm introducing local disturbances should be developed. This would also allow the evaluation of the compression system response to different stall inception mechanisms.
- A more rigorous derivation and further development of shear stress effects is essential since mixing is an important modelling aspect at forward flow but predominantly at reverse flow conditions. This will introduce a numerical mixing mechanism which will promote the formation of more uniform profiles downstream blade passages and prevent unnatural behaviour during highly unsteady, non-uniform flows.
- Further insight into the limitations and validity of the blockage-mixing model is required. The validity of the model could be evaluated on 2D axially and radially positioned cascades with a domain large enough to ensure fully mixed-out flow and no recirculation regions in the vicinity of the blade passage. Compressibility effects are essential in high-speed compressors and are neglected by the current model apart from the mixing loss, where a compressibility correction is applied.
- The definition of the mixing rate inside blade passages based on fundamental principles would offer a physics-based model which would intrinsically define the mixing rate inside the blade passage without this constituting a tuning parameter.
- Parametric, 2D, CFD studies on axially and radially positioned cascades could provide a link between geometrical blade features (stagger angle, wrap angle etc.) or flow quantities e.g. incidence angle and key reverse flow features e.g. aerodynamic blockage, losses deviation and mixing rate. The quantities of interest could then be expressed as a function of the aforementioned geometric and flow-field variables in the form of empirical correlations. The data could

also be used to assess the accuracy of the analytical blockage-mixing model and implement corrections factors.

- Further validation of the code is required. The code has the potential of predicting the pre and post-stall aerodynamic performance of axi-centrifugal compressors. This would be of particular interest since no through-flow codes capable of predicting pre and post-stall behaviour of axi-centrifugal compressors are reported in the literature.
- Parallelisation of the code could drastically accelerate the solution of the flow-field by enabling parallel processing, further decreasing computational time.

Bibliography

- [1] International Civil Aviation Organization, *Air transport, passengers carried*, 2022. [Online]. Available: {<https://data.worldbank.org/indicator/IS.AIR.PSGR?end=2020&start=1970>}.
- [2] International Civil Aviation Organization, *Aircraft Engine Emissions*, 2022. [Online]. Available: {<https://www.icao.int/environmental-protection/Pages/aircraft-engine-emissions.aspx>}.
- [3] J. Green, “Civil aviation and the environmental challenge,” *The aeronautical journal*, vol. 107, no. 1072, pp. 281–299, 2003.
- [4] I. C. A. Organization, *Aircraft Engine Emissions a Vision for 2020 Meeting society’s needs and winning global leadership*, 2001. [Online]. Available: {[http s://www.icao.int/environmental-protection/Pages/aircraft-engine-emissions.aspx](https://www.icao.int/environmental-protection/Pages/aircraft-engine-emissions.aspx)}.
- [5] E. Commission, D.-G. for Mobility, Transport, D.-G. for Research, and Innovation, *Flightpath 2050 : Europe’s vision for aviation : maintaining global leadership and serving society’s needs*. Publications Office, 2012. DOI: doi/10.2777/15458.
- [6] P. Rompokos, A. Rolt, D. Nalianda, A. T. Isikveren, C. Senné, T. Gronstedt, and H. Abedi, “Synergistic Technology Combinations for Future Commercial Aircraft Using Liquid Hydrogen,” *Journal of Engineering for Gas Turbines and Power*, vol. 143, no. 7, Mar. 2021, 071017, ISSN: 0742-4795. DOI: 10.1115/1.4049694. eprint: <https://asmedigitalcollection.asme.org/gasturbinespower/article-pdf/143/7/071017/6674143/gtp\143\07\071017.pdf>. [Online]. Available: <https://doi.org/10.1115/1.4049694>.
- [7] A. Guha, “Optimum fan pressure ratio for bypass engines with separate or mixed exhaust streams,” *Journal of Propulsion and Power*, vol. 17, no. 5, pp. 1117–1122, 2001.

- [8] P. Walsh and P. Fletcher, “Design point performance and engine concept design,” *Gas Turbine Performance, 2nd ed.*; Blackwell Science Ltd.: Oxford, UK, 2004.
- [9] D. G. Bogard and K. A. Thole, “Gas turbine film cooling,” *Journal of propulsion and power*, vol. 22, no. 2, pp. 249–270, 2006.
- [10] R. Van den Braembussche, *Design and Analysis of Centrifugal Compressors*. ASME Press, Jan. 2020, ISBN: 9781119424093. DOI: 10.1115/1.861VAN. [Online]. Available: <https://doi.org/10.1115/1.861VAN>.
- [11] GE Honda, *Explore the HF 120 Engine*, 2022. [Online]. Available: <https://www.gehonda.com/engine/explore.html>.
- [12] Wikipedia, *Rolls-Royce Turbomeca RTM322*, 2022. [Online]. Available: https://en.wikipedia.org/wiki/Rolls-Royce_Turbomeca_RTM322.
- [13] R. Von der Bank, S. Donnerhack, A. Rae, F. Poutriquet, A. Lundblad, A. Antoranz, L. Tarnowski, and M. Ruzicka, “Compressors for ultra-high-pressure-ratio aero-engines,” *CEAS Aeronautical Journal*, vol. 7, no. 3, pp. 455–470, 2016.
- [14] G. Medic, O. P. Sharma, J. Jongwook, L. W. Hardin, D. C. McCormick, W. T. Cousins, E. A. Lurie, A. Shabbir, B. M. Holley, and P. R. Van Slooten, “High efficiency centrifugal compressor for rotorcraft applications,” *NASA Technical Report*, no. NASA/CR—2014-218114/REV1, Oct. 2017.
- [15] S. Cline, W. Fesler, H. Liu, R. Lovell, and S. Shaffer, “Energy efficient engine—high pressure compressor component performance report,” *NASA Technical Report*, no. NASA CR-168245, Oct. 1983.
- [16] S. Dixon and C. Hall, “Fluid mechanics and thermodynamics of turbomachinery (seventh edition),” in S. Dixon and C. Hall, Eds., *Seventh Edition*, Boston: Butterworth-Heinemann, 2014, pp. 39–67, ISBN: 978-0-12-415954-9. DOI: <https://doi.org/10.1016/B978-0-12-415954-9.00002-4>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/B9780124159549000024>.
- [17] J. Anderson, *EBOOK: Fundamentals of Aerodynamics (SI units)*. McGraw Hill, 2011.

- [18] The Aviationist, *Watch an F-16 suffer a compressor stall during the display at AirPower 2016 airshow*, 2016. [Online]. Available: <https://theaviationist.com/2016/09/07/watch-an-f-16-suffer-a-compressor-stall-during-the-display-at-airpower-2016-airshow/>.
- [19] Airlines, *Boeing 777-35R/ER - Turkish Airlines*, 2010. [Online]. Available: <https://www.airliners.net/photo/Turkish-Airlines/Boeing-777-35R-ER/2026037/L>.
- [20] *Experimental Investigation of Centrifugal Compressor Stabilization Techniques*, vol. Volume 6: Turbo Expo 2003, Parts A and B, Turbo Expo: Power for Land, Sea, and Air, Jun. 2003, pp. 765–776. DOI: 10.1115/GT2003-38524. eprint: <https://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT2003/3/36894/765/4550270/765\1.pdf>. [Online]. Available: <https://doi.org/10.1115/GT2003-38524>.
- [21] J. Kurzke, “Effects of Inlet Flow Distortion on the Performance of Aircraft Gas Turbines,” *Journal of Engineering for Gas Turbines and Power*, vol. 130, no. 4, Apr. 2008, 041201, ISSN: 0742-4795. DOI: 10.1115/1.2901190. eprint: <https://asmedigitalcollection.asme.org/gasturbinespower/article-pdf/130/4/041201/5635187/041201\1.pdf>. [Online]. Available: <https://doi.org/10.1115/1.2901190>.
- [22] L. Harjes, C. Bode, J. Grubert, P. Frantzheld, P. Koch, and J. Friedrichs, “Investigation of jet engine intake distortions caused by crosswind conditions,” *Journal of the Global Power and Propulsion Society*, vol. 4, pp. 48–62, 2020. DOI: {10.33737/jgpps/118875}. [Online]. Available: <https://doi.org/10.33737/jgpps/118875>.
- [23] D. J. Rajendran and V. Pachidis, “Flow Distortion Into the Core Engine for an Installed Variable Pitch Fan in Reverse Thrust Mode,” *Journal of Turbomachinery*, vol. 143, no. 7, Apr. 2021, 071001, ISSN: 0889-504X. DOI: 10.1115/1.4050331. eprint: <https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/143/7/071001/6679455/turbo\143\7\071001.pdf>. [Online]. Available: <https://doi.org/10.1115/1.4050331>.
- [24] R. Kurz and K. Brun, “Degradation in Gas Turbine Systems,” *Journal of Engineering for Gas Turbines and Power*, vol. 123, no. 1, pp. 70–77, Nov. 2000, ISSN: 0742-4795. DOI: 10.1115/1.1340629. eprint: <https://asmedigitalcollection.asme.org/gasturbinespower/article-pdf/123/1/70/5746793/70\1.pdf>. [Online]. Available: <https://doi.org/10.1115/1.1340629>.

- [25] Z. S. Spakovszky, J. B. Gertz, O. P. Sharma, J. D. Paduano, A. H. Epstein, and E. M. Greitzer, “Influence of Compressor Deterioration on Engine Dynamic Behavior and Transient Stall-Margin,” *Journal of Turbomachinery*, vol. 122, no. 3, pp. 477–484, Feb. 1999, ISSN: 0889-504X. DOI: 10.1115/1.1303817. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/122/3/477/5515041/477_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.1303817>.
- [26] *Modelling the Transient Behaviour of Jet Engines*, vol. Volume 1: Aircraft Engine; Marine; Turbomachinery; Microturbines and Small Turbomachinery, Turbo Expo: Power for Land, Sea, and Air, V001T01A016, May 2000. DOI: 10.1115/2000-GT-0575. eprint: <https://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT2000/78545/V001T01A016/4222317/v001t01a016-2000-gt-0575.pdf>. [Online]. Available: <https://doi.org/10.1115/2000-GT-0575>.
- [27] M. Righi, “Three-Dimensional Through-Flow Modelling of Axial Compressor Rotating Stall and Surge,” PhD Thesis, Cranfield University, 2017.
- [28] I. J. Day, “Stall, Surge, and 75 Years of Research,” *Journal of Turbomachinery*, vol. 138, no. 1, Oct. 2015, 011001, ISSN: 0889-504X. DOI: 10.1115/1.4031473. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/138/1/011001/6303198/turbo_138_01_011001.pdf. [Online]. Available: <https://doi.org/10.1115/1.4031473>.
- [29] H. W. Emmons, C. E. Pearson, and H. P. Grant, “Compressor Surge and Stall Propagation,” *Transactions of the American Society of Mechanical Engineers*, vol. 77, no. 4, pp. 455–467, Jul. 2022, ISSN: 0097-6822. DOI: 10.1115/1.4014389. eprint: https://asmedigitalcollection.asme.org/fluidsengineering/article-pdf/77/4/455/6894807/455_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.4014389>.
- [30] M. Righi, V. Pachidis, and L. Könözy, “On the prediction of the reverse flow and rotating stall characteristics of high-speed axial compressors using a three-dimensional through-flow code,” *Aerospace Science and Technology*, vol. 99, p. 105578, 2020, ISSN: 1270-9638. DOI: <https://doi.org/10.1016/j.ast.2019.105578>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S1270963819320504>.

- [31] J. Dodds and M. Vahdati, “Rotating Stall Observations in a High Speed Compressor—Part II: Numerical Study,” *Journal of Turbomachinery*, vol. 137, no. 5, May 2015, 051003, ISSN: 0889-504X. DOI: 10.1115/1.4028558. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/137/5/051003/6302009/turbo_137_05_051003.pdf. [Online]. Available: <https://doi.org/10.1115/1.4028558>.
- [32] E. Munari, M. Morini, M. Pinelli, P. R. Spina, and A. Suman, “Experimental Investigation of Stall and Surge in a Multistage Compressor,” *Journal of Engineering for Gas Turbines and Power*, vol. 139, no. 2, Sep. 2016, 022605, ISSN: 0742-4795. DOI: 10.1115/1.4034239. eprint: https://asmedigitalcollection.asme.org/gasturbinespower/article-pdf/139/2/022605/6174913/gtp_139_02_022605.pdf. [Online]. Available: <https://doi.org/10.1115/1.4034239>.
- [33] K. H. Powers, I. J. Kennedy, C. J. Brace, P. A. Milewski, and C. D. Copeland, “Development and Validation of a Model for Centrifugal Compressors in Reversed Flow Regimes,” *Journal of Turbomachinery*, vol. 143, no. 10, May 2021, 101001, ISSN: 0889-504X. DOI: 10.1115/1.4050668. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/143/10/101001/6695641/turbo_143_10_101001.pdf. [Online]. Available: <https://doi.org/10.1115/1.4050668>.
- [34] M. Righi, V. Pachidis, L. Könözy, T. Giersch, and S. Schrape, “Experimental validation of a three-dimensional through-flow model for high-speed compressor surge,” *Aerospace Science and Technology*, vol. 128, p. 107775, 2022, ISSN: 1270-9638. DOI: <https://doi.org/10.1016/j.ast.2022.107775>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S1270963822004497>.
- [35] Z. S. Spakovszky and C. H. Roduner, “Spike and Modal Stall Inception in an Advanced Turbocharger Centrifugal Compressor,” *Journal of Turbomachinery*, vol. 131, no. 3, Apr. 2009, 031012, ISSN: 0889-504X. DOI: 10.1115/1.2988166. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/131/3/031012/5845661/031012_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.2988166>.
- [36] M. Righi, V. Pachidis, L. Könözy, T. Giersch, and S. Schrape, “Experimental validation of a three-dimensional through-flow model for high-speed compressor surge,” *Aerospace Science and Technology*, vol. 128, p. 107775, 2022, ISSN:

- 1270-9638. DOI: <https://doi.org/10.1016/j.ast.2022.107775>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S1270963822004497>.
- [37] F. Zhao, J. Dodds, and M. Vahdati, "Poststall Behavior of a Multistage High Speed Compressor at Off-Design Conditions," *Journal of Turbomachinery*, vol. 140, no. 12, Oct. 2018, 121002, ISSN: 0889-504X. DOI: 10.1115/1.4041142. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/140/12/121002/6310027/turbo_140_12_121002.pdf. [Online]. Available: <https://doi.org/10.1115/1.4041142>.
- [38] G. F. Kiwada, "Development of a body force description for compressor stability assessment," PhD Thesis, Massachusetts Institute of Technology, 2008.
- [39] *A Study of Spike and Modal Stall Phenomena in a Low-Speed Axial Compressor*, vol. Volume 1: Aircraft Engine; Marine; Turbomachinery; Microturbines and Small Turbomachinery, Turbo Expo: Power for Land, Sea, and Air, V001T03A109, Jun. 1997. DOI: 10.1115/97-GT-526. eprint: <https://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT1997/78682/V001T03A109/2408802/v001t03a109-97-gt-526.pdf>. [Online]. Available: <https://doi.org/10.1115/97-GT-526>.
- [40] J. P. Longley, "A Review of Nonsteady Flow Models for Compressor Stability," *Journal of Turbomachinery*, vol. 116, no. 2, pp. 202–215, Apr. 1994, ISSN: 0889-504X. DOI: 10.1115/1.2928354. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/116/2/202/5629136/202_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.2928354>.
- [41] J. Paduano, E. Greitzer, and A. Epstein, "Compression system stability and active control," *Annual Review of Fluid Mechanics*, vol. 33, no. 1, pp. 491–517, 2001. DOI: 10.1146/annurev.fluid.33.1.491. eprint: <https://doi.org/10.1146/annurev.fluid.33.1.491>. [Online]. Available: <https://doi.org/10.1146/annurev.fluid.33.1.491>.
- [42] F. K. Moore and E. M. Greitzer, "A Theory of Post-Stall Transients in Axial Compression Systems: Part I—Development of Equations," *Journal of Engineering for Gas Turbines and Power*, vol. 108, no. 1, pp. 68–76, Jan. 1986, ISSN: 0742-4795. DOI: 10.1115/1.3239887. eprint: https://asmedigitalcollection.asme.org/gasturbinespower/article-pdf/108/1/68/5567788/68_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.3239887>.

- [43] N. M. McDougall, N. A. Cumpsty, and T. P. Hynes, "Stall Inception in Axial Compressors," *Journal of Turbomachinery*, vol. 112, no. 1, pp. 116–123, Jan. 1990, ISSN: 0889-504X. DOI: 10.1115/1.2927406. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/112/1/116/5839673/116_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.2927406>.
- [44] G. Pullan, A. M. Young, I. J. Day, E. M. Greitzer, and Z. S. Spakovszky, "Origins and Structure of Spike-Type Rotating Stall," *Journal of Turbomachinery*, vol. 137, no. 5, May 2015, 051007, ISSN: 0889-504X. DOI: 10.1115/1.4028494. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/137/5/051007/6301708/turbo_137_05_051007.pdf. [Online]. Available: <https://doi.org/10.1115/1.4028494>.
- [45] H. Park, "Unsteady disturbance structures in axial flow compressor stall inception," Ph.D. dissertation, Massachusetts Institute of Technology, 1994.
- [46] I. J. Day, "Stall Inception in Axial Flow Compressors," *Journal of Turbomachinery*, vol. 115, no. 1, pp. 1–9, Jan. 1993, ISSN: 0889-504X. DOI: 10.1115/1.2929209. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/115/1/1/5584592/1_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.2929209>.
- [47] C. Tan, I Day, S Morris, and A Wadia, "Spike-type compressor stall inception, detection, and control," *Annual review of fluid mechanics*, vol. 42, pp. 275–300, 2010.
- [48] J. N. Everitt and Z. S. Spakovszky, "An Investigation of Stall Inception in Centrifugal Compressor Vaned Diffuser1," *Journal of Turbomachinery*, vol. 135, no. 1, Nov. 2012, 011025, ISSN: 0889-504X. DOI: 10.1115/1.4006533. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/135/1/011025/6294598/turb_135_1_011025.pdf. [Online]. Available: <https://doi.org/10.1115/1.4006533>.
- [49] B. Benneke, "A methodology for centrifugal compressor stability prediction," MSc Thesis, Massachusetts Institute of Technology, 2009.
- [50] Z. S. Spakovszky, "Backward Traveling Rotating Stall Waves in Centrifugal Compressors," *Journal of Turbomachinery*, vol. 126, no. 1, pp. 1–12, Mar. 2004, ISSN: 0889-504X. DOI: 10.1115/1.1643382. eprint: <https://asmedigi>

- talcollection.asme.org/turbomachinery/article-pdf/126/1/1/5634509/1_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.1643382>.
- [51] T. Iura and W. D. Rannie, “Experimental Investigations of Propagating Stall in Axial-Flow Compressors,” *Transactions of the American Society of Mechanical Engineers*, vol. 76, no. 3, pp. 463–471, Jul. 2022, ISSN: 0097-6822. DOI: 10.1115/1.4014879. eprint: https://asmedigitalcollection.asme.org/fluidsengineering/article-pdf/76/3/463/6893216/463_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.4014879>.
- [52] E. M. Greitzer, “Review—Axial Compressor Stall Phenomena,” *Journal of Fluids Engineering*, vol. 102, no. 2, pp. 134–151, Jun. 1980, ISSN: 0098-2202. DOI: 10.1115/1.3240634. eprint: https://asmedigitalcollection.asme.org/fluidsengineering/article-pdf/102/2/134/5700713/134_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.3240634>.
- [53] I. J. Day and N. A. Cumpsty, “The measurement and interpretation of flow within rotating stall cells in axial compressors,” *Archive: Journal of Mechanical Engineering Science 1959-1982 (vols 1-23)*, vol. 20, pp. 101–114, 1978.
- [54] R. Van den Braembussche, *Design and Analysis of Centrifugal Compressors*. ASME Press, Jan. 2020, ISBN: 9781119424093. DOI: 10.1115/1.861VAN. [Online]. Available: <https://doi.org/10.1115/1.861VAN>.
- [55] P. Frigne and R. Van Den Braembussche, “Distinction Between Different Types of Impeller and Diffuser Rotating Stall in a Centrifugal Compressor With Vaneless Diffuser,” *Journal of Engineering for Gas Turbines and Power*, vol. 106, no. 2, pp. 468–474, Apr. 1984, ISSN: 0742-4795. DOI: 10.1115/1.3239589. eprint: https://asmedigitalcollection.asme.org/gasturbinespower/article-pdf/106/2/468/5677153/468_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.3239589>.
- [56] W. Jansen, “Rotating Stall in a Radial Vaneless Diffuser,” *Journal of Basic Engineering*, vol. 86, no. 4, pp. 750–758, Dec. 1964, ISSN: 0021-9223. DOI: 10.1115/1.3655945. eprint: https://asmedigitalcollection.asme.org/fluidsengineering/article-pdf/86/4/750/5764391/750_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.3655945>.
- [57] W. Oakes, P. Lawless, and S. Fleeter, “Characterization of the behavior of a centrifugal compressor with active stall control,” in *31st Joint Propulsion Conference and Exhibit*, 1995, p. 2484.

- [58] Righi, Mauro and Pachidis, Vassilios and Könözy, László and Zhao, Fanzhou and Vahdati, Mehdi, “Three-dimensional low-order surge model for high-speed axial compressors,” *Journal of the Global Power and Propulsion Society*, vol. 4, 274–284, 2020. DOI: {10.33737/jgpps/130790}. [Online]. Available: {<https://doi.org/10.33737/jgpps/130790>}.
- [59] R. N. Gamache and E. M. Greitzer, “Reverse flow in multistage axial compressors,” *Journal of Propulsion and Power*, vol. 6, no. 4, pp. 461–473, 1990. DOI: 10.2514/3.25458. eprint: <https://doi.org/10.2514/3.25458>. [Online]. Available: <https://doi.org/10.2514/3.25458>.
- [60] I. J. Day, “Axial compressor performance during surge,” *Journal of Propulsion and Power*, vol. 10, no. 3, pp. 329–336, 1994. DOI: 10.2514/3.23760. eprint: <https://doi.org/10.2514/3.23760>. [Online]. Available: <https://doi.org/10.2514/3.23760>.
- [61] M. P. Wernet, M. M. Bright, and G. J. Skoch, “An Investigation of Surge in a High-Speed Centrifugal Compressor Using Digital PIV,” *Journal of Turbomachinery*, vol. 123, no. 2, pp. 418–428, Jan. 2000, ISSN: 0889-504X. DOI: 10.1115/1.1343465. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/123/2/418/5601644/418_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.1343465>.
- [62] M. P. Wernet, M. M. Bright, and G. J. Skoch, “An Investigation of Surge in a High-Speed Centrifugal Compressor Using Digital PIV,” *NASA Technical Report*, no. NASA/TM-2002-211832, Dec. 2002.
- [63] K. H. Powers, I. J. Kennedy, C. J. Brace, P. A. Milewski, and C. D. Copeland, “Development and Validation of a Model for Centrifugal Compressors in Reversed Flow Regimes,” *Journal of Turbomachinery*, vol. 143, no. 10, May 2021, 101001, ISSN: 0889-504X. DOI: 10.1115/1.4050668. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/143/10/101001/6695641/turbo_143_10_101001.pdf. [Online]. Available: <https://doi.org/10.1115/1.4050668>.
- [64] R. Aungier, *Axial-flow Compressors: A Strategy for Aerodynamic Design and Analysis* (EngineeringPro collection). ASME Press, 2003, ISBN: 9781860584220. [Online]. Available: <https://books.google.co.uk/books?id=E91SAAAAMAAJ>.

- [65] A. Mikhailov, A. Mikhailova, and D. Akhmedzyanov, “New 1-D Method for the Prediction of Axial-flow Compressors Off-design Performance,” *Procedia Engineering*, vol. 150, pp. 155–160, 2016, 2nd International Conference on Industrial Engineering (ICIE-2016), ISSN: 1877-7058. DOI: <https://doi.org/10.1016/j.proeng.2016.06.740>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S1877705816312899>.
- [66] H Oh, E. Yoon, and M. Chung, “An optimum set of loss models for performance prediction of centrifugal compressors,” *Proceedings of The Institution of Mechanical Engineers Part A-journal of Power and Energy - PROC INST MECH ENG A-J POWER*, vol. 211, pp. 331–338, Jun. 1997. DOI: 10.1243/0957650971537231.
- [67] M. R. Galvas, “A compressor designed for the energy research and development agency automotive gas turbine program,” *NASA Technical Report*, no. NASA-TM-X-71719, May 1975.
- [68] H Ronald, “Centrifugal compressors: A strategy for aerodynamic design and analysis,” *American Society of Mechanical Engineers Press*, 2000.
- [69] E. I. GUTIÉRREZ VELÁSQUEZ, “Determination of a suitable set of loss models for centrifugal compressor performance prediction,” *Chinese Journal of Aeronautics*, vol. 30, no. 5, pp. 1644–1650, 2017, ISSN: 1000-9361. DOI: <https://doi.org/10.1016/j.cja.2017.08.002>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S1000936117301772>.
- [70] P Harley, S Spence, J Early, D Filsinger, and M Dietrich, “An evaluation of 1d loss model collections for the off-design performance prediction of automotive turbocharger compressors,” *IOP Conference Series: Materials Science and Engineering*, vol. 52, no. 4, p. 042011, 2013. DOI: 10.1088/1757-899X/52/4/042011. [Online]. Available: <https://dx.doi.org/10.1088/1757-899X/52/4/042011>.
- [71] B. Kerres, S. Sanz, E. Sundström, and M. Mihaescu, “A comparison of performance predictions between 1d models and numerical data for a turbocharger compressor,” Jan. 2017. DOI: 10.29008/ETC2017-350.
- [72] E. M. Greitzer, “Surge and Rotating Stall in Axial Flow Compressors—Part I: Theoretical Compression System Model,” *Journal of Engineering for Power*, vol. 98, no. 2, pp. 190–198, Apr. 1976, ISSN: 0022-0825. DOI: 10.1115/1.3446138. eprint: <https://asmedigitalcollection.asme.org/gasturbines>

- power/article-pdf/98/2/190/5886245/190_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.3446138>.
- [73] E. M. Greitzer, “Surge and Rotating Stall in Axial Flow Compressors—Part II: Experimental Results and Comparison With Theory,” *Journal of Engineering for Power*, vol. 98, no. 2, pp. 199–211, Apr. 1976, ISSN: 0022-0825. DOI: 10.1115/1.3446139. eprint: https://asmedigitalcollection.asme.org/gasturbinespower/article-pdf/98/2/199/5886325/199_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.3446139>.
- [74] E. M. Greitzer and F. K. Moore, “A Theory of Post-Stall Transients in Axial Compression Systems: Part II—Application,” *Journal of Engineering for Gas Turbines and Power*, vol. 108, no. 2, pp. 231–239, Apr. 1986, ISSN: 0742-4795. DOI: 10.1115/1.3239893. eprint: https://asmedigitalcollection.asme.org/gasturbinespower/article-pdf/108/2/231/5750227/231_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.3239893>.
- [75] K. H. Powers, C. J. Brace, C. J. Budd, C. D. Copeland, and P. A. Milewski, “Modeling Axisymmetric Centrifugal Compressor Characteristics From First Principles,” *Journal of Turbomachinery*, vol. 142, no. 9, Aug. 2020, 091010, ISSN: 0889-504X. DOI: 10.1115/1.4047616. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/142/9/091010/6559921/turbo_142_9_091010.pdf. [Online]. Available: <https://doi.org/10.1115/1.4047616>.
- [76] F. Marble, *Aerodynamics of Turbines and Compressors*, W. R. Hawthorne, Ed. Princeton: Princeton University Press, 2017, ISBN: 9781400885985. DOI: doi:10.1515/9781400885985. [Online]. Available: <https://doi.org/10.1515/9781400885985>.
- [77] J. D. Denton, “Throughflow Calculations for Transonic Axial Flow Turbines,” *Journal of Engineering for Power*, vol. 100, no. 2, pp. 212–218, Apr. 1978, ISSN: 0022-0825. DOI: 10.1115/1.3446336. eprint: https://asmedigitalcollection.asme.org/gasturbinespower/article-pdf/100/2/212/5560452/212_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.3446336>.
- [78] I. Templalex, P. Pilidis, V. Pachidis, and P. Kotsiopoulos, “Development of a Two-Dimensional Streamline Curvature Code,” *Journal of Turbomachinery*, vol. 133, no. 1, Sep. 2010, 011003, ISSN: 0889-504X. DOI: 10.1115/1.2720877. eprint: <https://asmedigitalcollection.asme.org/turbomachinery/ar>

- ticle-pdf/133/1/011003/5848655/011003_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.2720877>.
- [79] V. Pachidis, P. Pilidis, I. Templalexis, T. Korakianitis, and P. Kotsiopoulos, “Prediction of Engine Performance Under Compressor Inlet Flow Distortion Using Streamline Curvature,” *Journal of Engineering for Gas Turbines and Power*, vol. 129, no. 1, pp. 97–103, Feb. 2006, ISSN: 0742-4795. DOI: 10.1115/1.2363414. eprint: https://asmedigitalcollection.asme.org/gasturbinespower/article-pdf/129/1/97/5892685/97_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.2363414>.
- [80] *An Improved Streamline Curvature Method for Centrifugal Compressor Performance*, vol. Volume 1: Compressors, Fans and Pumps; Turbines; Heat Transfer; Combustion, Fuels and Emissions, Gas Turbine India Conference, Dec. 2017. DOI: 10.1115/GTINDIA2017-4531. eprint: <https://asmedigitalcollection.asme.org/GTINDIA/proceedings-pdf/GTINDIA2017/58509/V001T01A005/2439531/v001t01a005-gtindia2017-4531.pdf>. [Online]. Available: <https://doi.org/10.1115/GTINDIA2017-4531>.
- [81] *An Analytical Shock-Loss Model Fully Integrated to a Streamline Curvature Method for Axial-Flow Compressors*, vol. Volume 2C: Turbomachinery, Turbo Expo: Power for Land, Sea, and Air, Jun. 2019. DOI: 10.1115/GT2019-91701. eprint: <https://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT2019/58578/V02CT41A036/6438710/v02ct41a036-gt2019-91701.pdf>. [Online]. Available: <https://doi.org/10.1115/GT2019-91701>.
- [82] J. Escuret and V. Garnier, “Numerical simulations of surge and rotating-stall in multi-stage axial-flow compressors,” in *30th Joint Propulsion Conference and Exhibit*. DOI: 10.2514/6.1994-3202. eprint: <https://arc.aiaa.org/doi/pdf/10.2514/6.1994-3202>. [Online]. Available: <https://arc.aiaa.org/doi/abs/10.2514/6.1994-3202>.
- [83] *Axisymmetric Aerodynamic Numerical Analysis of a Turbofan Engine*, vol. Volume 2: Aircraft Engine; Marine; Microturbines and Small Turbomachinery, Turbo Expo: Power for Land, Sea, and Air, V002T02A013, Jun. 1995. DOI: 10.1115/95-GT-338. eprint: <https://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT1995/78798/V002T02A013/4457186/v002t02a013-95-gt-338.pdf>. [Online]. Available: <https://doi.org/10.1115/95-GT-338>.

- [84] S. Zoppellari, “Analytical Modeling of Rotating Stall and Surge,” PhD Thesis, Cranfield University, 2014.
- [85] A. P. Kottapalli, “Development of a body force model for centrifugal compressors,” MSc Thesis, Massachusetts Institute of Technology, 2013.
- [86] Y. Gong, “A computational model for rotating stall and inlet distortions in multistage compressors,” PhD Thesis, Massachusetts Institute of Technology, 1999.
- [87] E. Hsiao, M. Naimi, J. P. Lewis, K. Dalbey, Y. Gong, and C. Tan, “Actuator duct model of turbomachinery components for powered-nacelle navier-stokes calculations,” *Journal of Propulsion and Power*, vol. 17, no. 4, pp. 919–927, 2001. DOI: 10.2514/2.5825. eprint: <https://doi.org/10.2514/2.5825>. [Online]. Available: <https://doi.org/10.2514/2.5825>.
- [88] M. L. Brand, “An improved blade passage model for estimating off-design axial compressor performance,” MSc Thesis, Massachusetts Institute of Technology, 2013.
- [89] J. Kerner, “An Assessment of Body Force Representations for Compressor Stall Simulation,” MSc Thesis, Massachusetts Institute of Technology, 2010.
- [90] *A Three-Dimensional Unsteady CFD Model of Compressor Stability*, vol. Volume 6: Turbomachinery, Parts A and B, Turbo Expo: Power for Land, Sea, and Air, May 2006, pp. 1157–1168. DOI: 10.1115/GT2006-90040. eprint: <https://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT2006/4241X/1157/4532714/1157\1.pdf>. [Online]. Available: <https://doi.org/10.1115/GT2006-90040>.
- [91] *Calculating Stall and Surge Transients*, vol. Volume 6: Turbo Expo 2007, Parts A and B, Turbo Expo: Power for Land, Sea, and Air, May 2007, pp. 125–136. DOI: 10.1115/GT2007-27378. eprint: <https://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT2007/47950/125/2677621/125\1.pdf>. [Online]. Available: <https://doi.org/10.1115/GT2007-27378>.
- [92] M. Righi, V. Pachidis, L. Könözy, and L. Pawsey, “Three-dimensional through-flow modelling of axial flow compressor rotating stall and surge,” *Aerospace Science and Technology*, vol. 78, pp. 271–279, 2018, ISSN: 1270-9638. DOI: <https://doi.org/10.1016/j.ast.2018.04.021>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S1270963817322794>.

- [93] M. Righi, V. Pachidis, L. Könözy, T. Giersch, and S. Schrape, “Experimental validation of a three-dimensional through-flow model for high-speed compressor surge,” *Aerospace Science and Technology*, vol. 128, p. 107 775, 2022, ISSN: 1270-9638. DOI: <https://doi.org/10.1016/j.ast.2022.107775>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S1270963822004497>.
- [94] W. Thollet, “Body force modeling of fan-airframe interactions,” PhD Thesis, Institut Supérieur de l’Aéronautique et de l’Espace, 2017.
- [95] E. Fadlun, R. Verzicco, P. Orlandi, and J. Mohd-Yusof, “Combined Immersed-Boundary Finite-Difference Methods for Three-Dimensional Complex Flow Simulations,” *Journal of Computational Physics*, vol. 161, no. 1, pp. 35–60, 2000, ISSN: 0021-9991. DOI: <https://doi.org/10.1006/jcph.2000.6484>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S0021999100964842>.
- [96] T. Cao, P. Hield, and P. G. Tucker, “Hierarchical Immersed Boundary Method with Smeared Geometry,” *Journal of Propulsion and Power*, vol. 33, no. 5, pp. 1151–1163, 2017. DOI: 10.2514/1.B36190. [Online]. Available: <https://doi.org/10.2514/1.B36190>.
- [97] A. A. Patel, “Assessment of a body force representation for compressor stability estimation,” MSc Thesis, Massachusetts Institute of Technology, 2009.
- [98] Y. Gong, C. S. Tan, K. A. Gordon, and E. M. Greitzer, “A Computational Model for Short-Wavelength Stall Inception and Development in Multistage Compressors,” *Journal of Turbomachinery*, vol. 121, no. 4, pp. 726–734, Oct. 1999, ISSN: 0889-504X. DOI: 10.1115/1.2836726. eprint: <https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/121/4/726/5619582/726\1.pdf>. [Online]. Available: <https://doi.org/10.1115/1.2836726>.
- [99] A. Peters, “Ultra-Short Nacelles for Low Fan Pressure Ratio Propulsors,” PhD Thesis, Massachusetts Institute of Technology, 2014.
- [100] H. L. Moses and S. B. Thomason, “An approximation for fully stalled cascades,” *Journal of Propulsion and Power*, vol. 2, no. 2, pp. 188–189, 1986. DOI: 10.2514/3.22865. eprint: <https://doi.org/10.2514/3.22865>. [Online]. Available: <https://doi.org/10.2514/3.22865>.

- [101] D. K. Hall, “Analysis of Civil Aircraft Propulsors with Boundary Layer Ingestion,” PhD Thesis, Massachusetts Institute of Technology, 2015.
- [102] E. M. Greitzer, C. S. Tan, and M. B. Graf, *Internal Flow: Concepts and Applications* (Cambridge Engine Technology Series). Cambridge University Press, 2004. DOI: 10.1017/CB09780511616709.
- [103] W. K. Sorensen, “A body force model for cavitating inducers in rocket engine turbopumps,” MSc Thesis, Massachusetts Institute of Technology, 2014.
- [104] K. H. Powers, C. J. Brace, C. J. Budd, C. D. Copeland, and P. A. Milewski, “Modeling Axisymmetric Centrifugal Compressor Characteristics From First Principles,” *Journal of Turbomachinery*, vol. 142, no. 9, Aug. 2020, 091010, ISSN: 0889-504X. DOI: 10.1115/1.4047616. eprint: https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/142/9/091010/6559921/turbo_142_9_091010.pdf. [Online]. Available: <https://doi.org/10.1115/1.4047616>.
- [105] A. Inc., *Ansys fluent theory guide, 2019r2*, 2019.
- [106] I. Ansys and I. ANSYS, “Cfd user’s manual,” *Ansys Inc*, p. 54, 2019.
- [107] *Investigations of the Flow Through a High Pressure Ratio Centrifugal Impeller*, vol. Volume 5: Turbo Expo 2002, Parts A and B, Turbo Expo: Power for Land, Sea, and Air, Jun. 2002, pp. 649–657. DOI: 10.1115/GT2002-30394. eprint: https://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT2002/3610X/649/2575287/649_1.pdf. [Online]. Available: <https://doi.org/10.1115/GT2002-30394>.
- [108] E. Toro, *Riemann Solvers and Numerical Methods for Fluid Dynamics: A Practical Introduction*. Springer Berlin Heidelberg, 2009, ISBN: 9783540498346. [Online]. Available: <https://books.google.co.uk/books?id=SqEjX0um8o0C>.
- [109] J. Anderson, *EBOOK: Fundamentals of Aerodynamics (SI units)*. McGraw Hill, 2011.
- [110] B Einfeldt, C. Munz, P. Roe, and B Sjögreen, “On godunov-type methods near low densities,” *Journal of Computational Physics*, vol. 92, no. 2, pp. 273–295, 1991, ISSN: 0021-9991. DOI: [https://doi.org/10.1016/0021-9991\(91\)90211-3](https://doi.org/10.1016/0021-9991(91)90211-3). [Online]. Available: <https://www.sciencedirect.com/science/article/pii/0021999191902113>.

- [111] S. F. Davis, "Simplified second-order godunov-type methods," *SIAM Journal on Scientific and Statistical Computing*, vol. 9, no. 3, pp. 445–473, 1988. DOI: 10.1137/0909030. eprint: <https://doi.org/10.1137/0909030>. [Online]. Available: <https://doi.org/10.1137/0909030>.
- [112] B. Einfeldt, C.-D. Munz, P. L. Roe, and B. Sjögreen, "On godunov-type methods near low densities," *Journal of Computational Physics*, vol. 92, pp. 273–295, 1991.
- [113] M. W. Johnson and J. Moore, "The Development of Wake Flow in a Centrifugal Impeller," *Journal of Engineering for Power*, vol. 102, no. 2, pp. 382–389, Apr. 1980, ISSN: 0022-0825. DOI: 10.1115/1.3230265. eprint: https://asmedigitalcollection.asme.org/gasturbinespower/article-pdf/102/2/382/5639072/382_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.3230265>.
- [114] W. B. Briggs, "Effect of Mach Number on the Flow and Application of Compressibility Corrections in a Two-Dimensional Subsonic-Transonic Compressor Cascade Having Varied Porous-Wall Suction at the Blade Tips," *NASA Technical Report*, no. NACA-TN-2649, Mar. 1952.
- [115] M. Boyce, *Centrifugal Compressors: A Basic Guide* (EngineeringPro collection). PennWell, 2003, ISBN: 9780878148011. [Online]. Available: <https://books.google.co.uk/books?id=Euo81Pjuj-YC>.
- [116] C. Zhang, X. Dong, X. Liu, Z. Sun, S. Wu, Q. Gao, and C. Tan, "A method to select loss correlations for centrifugal compressor performance prediction," *Aerospace Science and Technology*, vol. 93, p. 105335, 2019, ISSN: 1270-9638. DOI: <https://doi.org/10.1016/j.ast.2019.105335>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S1270963819313616>.
- [117] *Evaluation and Enhancement of a One-Dimensional Performance Analysis Method for Centrifugal Compressors*, vol. Volume 2D: Turbomachinery, Turbo Expo: Power for Land, Sea, and Air, V02DT42A005, Jun. 2014. DOI: 10.1115/GT2014-25141. eprint: <https://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT2014/45639/V02DT42A005/4230555/v02dt42a005-gt2014-25141.pdf>. [Online]. Available: <https://doi.org/10.1115/GT2014-25141>.

- [118] *Analysis and Validation of a Unified Slip Factor Model for Impellers at Design and Off-Design Conditions*, vol. Volume 7: Turbomachinery, Parts A, B, and C, Turbo Expo: Power for Land, Sea, and Air, Jun. 2010, pp. 1711–1721. DOI: 10.1115/GT2010-22164. eprint: https://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT2010/44021/1711/4718571/1711_1.pdf. [Online]. Available: <https://doi.org/10.1115/GT2010-22164>.
- [119] T. F. McKain and G. J. Holbrook, “Coordinates for a High Performance 4:1 Pressure Ratio Centrifugal Compressor,” *NASA Technical Report*, no. NASA-CR-204134, Jul. 1997.
- [120] D. C. Urasek, W. T. Gorrell, and W. S. Cunnann, “Performance of two-stage fan having low-aspect-ratio first-stage rotor blading,” *NASA Technical Report*, no. NASA-TP-1493, Aug. 1979.
- [121] L. Reid and R. D. Moore, “Design and overall performance of four highly loaded, high speed inlet stages for an advanced high-pressure-ratio core compressor,” *NASA Technical Report*, no. NASA-TP-1337, Oct. 1978.
- [122] X. He and X. Zheng, “Performance improvement of transonic centrifugal compressors by optimization of complex three-dimensional features,” *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, vol. 231, no. 14, pp. 2723–2738, 2017. DOI: 10.1177/0954410016673395. eprint: <https://doi.org/10.1177/0954410016673395>. [Online]. Available: <https://doi.org/10.1177/0954410016673395>.
- [123] *Optimization of 6.2:1 Pressure Ratio Centrifugal Compressor Impeller by 3D Inverse Design*, vol. Volume 7: Turbomachinery, Parts A, B, and C, Turbo Expo: Power for Land, Sea, and Air, Jun. 2011, pp. 2167–2177. DOI: 10.1115/GT2011-46505. eprint: https://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT2011/54679/2167/2761150/2167_1.pdf. [Online]. Available: <https://doi.org/10.1115/GT2011-46505>.
- [124] K. Ekradi and A. Madadi, “Performance improvement of a transonic centrifugal compressor impeller with splitter blade by three-dimensional optimization,” *Energy*, vol. 201, p. 117582, 2020, ISSN: 0360-5442. DOI: <https://doi.org/10.1016/j.energy.2020.117582>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S0360544220306897>.

- [125] S. Kulkarni, T. A. Beach, and G. J. Skoch, “Computational study of the cc3 impeller and vaneless diffuser experiment,” in *49th AIAA/ASME/SAE/ASEE Joint Propulsion Conference*. DOI: 10.2514/6.2013-3631. eprint: <https://arc.aiaa.org/doi/pdf/10.2514/6.2013-3631>. [Online]. Available: <https://arc.aiaa.org/doi/abs/10.2514/6.2013-3631>.
- [126] B. J. Cantwell, “Fundamentals of compressible flow,” *Department of Aeronautics and Astronautics*, 1996.

Appendix A

Influence coefficient analysis for pressure gradient term

To eliminate the pressure gradient term, six equations are first reformulated in their differential form and subsequently combined to express the pressure gradient term as a function of other flow-field variables.

Meridional Velocity:

$$V_m = \cos\beta \ W$$

$$\boxed{dV_m = \cos\beta \ dW - \sin\beta \ d\beta \ W} \quad (\text{A.1})$$

Meridional Area:

$$\boxed{\frac{dA_m}{A_m} = \frac{dr}{r}} \quad (\text{A.2})$$

Continuity:

$$\dot{m} = \rho V_m \cos\beta \ A_m$$

$$d\dot{m} = d(\rho V_m \ A_m) = V_m \ A_m \ d\rho + \rho V_m dA_m + \rho A_m d(V_m) = 0$$

$$\frac{d\rho}{\rho} + \frac{dA_m}{A_m} + \frac{\cos\beta \ dW - \sin\beta \ d\beta \ W}{\cos\beta \ W} = 0$$

Due to the sign convention used within the context of the present work, the flow angle, β , is defined as negative, therefore $\sin\beta$, $\tan\beta$ and $d\beta$ are multiplied with (-1) .

$$\boxed{\frac{d\rho}{\rho} + \frac{dr}{r} + \frac{dW}{W} - \tan\beta \, d\beta = 0} \quad (\text{A.3})$$

Equation of State:

$$P = \rho RT$$

$$dp = RT \, d\rho + \rho R \, dT + \overbrace{\rho T \, dR}^0$$

Dividing by ρRT :

$$\boxed{\frac{dp}{p} = \frac{d\rho}{\rho} + \frac{dT}{T}} \quad (\text{A.4})$$

Gibbs equation:

$$Tds = dh - \frac{dP}{\rho}$$

Substituting:

$$dh = c_p T$$

$$\rho = \frac{P}{RT} = c_p \frac{(\gamma - 1)}{\gamma} \frac{P}{T}$$

The differential form of Gibbs equation is given by:

$$\boxed{\frac{ds}{c_p} = \frac{dT}{T} - \frac{(\gamma - 1)}{\gamma} \frac{dP}{P}} \quad (\text{A.5})$$

Viscous Force (Marble):

$$f_{p,visc} = T \frac{\partial s}{\partial l} = T \frac{\partial s}{\partial m} \cos\beta$$

Conservation of Rothalpy:

Rothalpy using the relative stagnation enthalpy is defined as:

$$I = h_{0,rel} - \frac{1}{2} \Omega^2 r^2$$

Differentiating:

$$dI = dh_{0,rel} - d\left(\frac{1}{2} \Omega^2 r^2\right) = dh_{0,rel} - (r^2 \Omega d\Omega + \Omega^2 r dr)$$

Rothalpy is conserved along a relative streamline, $dI = 0$ and the rotational speed is constant $d\Omega = 0$:

$$0 = dh_{0,rel} - \Omega^2 r dr$$

$$\Omega^2 r dr = c_p dT_{0,rel}$$

Dividing by $T_{0,rel}$:

$$\frac{\Omega^2 r dr}{T_{0,rel}} = c_p \frac{dT_{0,rel}}{T_{0,rel}}$$

The total relative temperature gradient is replaced with the following expression:

$$\frac{dT_{0,rel}}{T_{0,rel}} = \frac{dT}{T} \left[\frac{1}{1 + \frac{\gamma-1}{2} Ma_{rel}^2} \right] + \frac{dw}{w} \left[\frac{(\gamma-1) Ma_{rel}^2}{1 + \frac{\gamma-1}{2} Ma_{rel}^2} \right]$$

Substituting the above equation in the rothalpy expression above gives:

$$\begin{aligned} \frac{\Omega^2 r dr}{T_{0,rel}} &= c_p \frac{dT}{T} \left[\frac{1}{1 + \frac{\gamma-1}{2} Ma_{rel}^2} \right] + c_p \frac{dw}{w} \left[\frac{(\gamma-1) Ma_{rel}^2}{1 + \frac{\gamma-1}{2} Ma_{rel}^2} \right] \\ \frac{\left(1 + \frac{\gamma-1}{2} Ma_{rel}^2\right) u_\Omega^2 dr}{T_{0,rel} r} &= c_p \frac{dT}{T} [1] + c_p \frac{dw}{w} [(\gamma-1) Ma_{rel}^2] \end{aligned}$$

Using the following expression to convert rotational speed to its equivalent Mach number and expressing c_p in terms of γ :

$$\frac{u^2}{\gamma RT} = Ma^2$$

$$c_p = \frac{\gamma R}{\gamma - 1}$$

Substituting the above expressions gives:

$$\frac{\overbrace{\left(1 + \frac{\gamma - 1}{2} Ma_{rel}^2\right)}^{1/T}}{T_{0,rel}} \gamma R T Ma_{\Omega}^2 \frac{dr}{r} = \frac{\gamma R}{\gamma - 1} \frac{dT}{T} [1] + \frac{\gamma R}{\gamma - 1} \frac{dw}{w} [(\gamma - 1) Ma_{rel}^2]$$

Rearranging and collecting terms gives the differential form of rothalpy:

$$\boxed{\frac{dT}{T} [1] = \frac{dr}{r} [(\gamma - 1) Ma_{\Omega}^2] + \frac{dw}{w} [-(\gamma - 1) Ma_{rel}^2]} \quad (A.6)$$

Expression of pressure gradient term as function of other flow-field variables

Starting with rothalpy and substituting relative velocity term using continuity (eq.(A.3)):

$$\begin{aligned} \frac{dT}{T} &= \frac{dr}{r} [(\gamma - 1) Ma_{\Omega}^2] + \frac{dw}{w} [-(\gamma - 1) Ma_{rel}^2] \\ \frac{dT}{T} &= \frac{dr}{r} [(\gamma - 1) Ma_{\Omega}^2] + \left(-\frac{d\rho}{\rho} - \frac{dr}{r} + \tan\beta d\beta\right) [-(\gamma - 1) Ma_{rel}^2] \end{aligned}$$

Collecting terms:

$$\frac{dT}{T} = \frac{dr}{r} [(\gamma - 1) (Ma_{\Omega}^2 + Ma_{rel}^2)] + \left(-\frac{d\rho}{\rho} + \tan\beta d\beta\right) [-(\gamma - 1) Ma_{rel}^2]$$

Using equation of state to replace density terms (eq.(A.4)):

$$\frac{dT}{T} = \frac{dr}{r} [(\gamma - 1) (Ma_{\Omega}^2 + Ma_{rel}^2)] + \left(-\frac{dP}{P} + \frac{dT}{T} + \tan\beta d\beta\right) [-(\gamma - 1) Ma_{rel}^2]$$

Collecting terms:

$$\frac{dT}{T} [1 + (\gamma - 1) Ma_{rel}^2] = \frac{dr}{r} [(\gamma - 1) (Ma_{\Omega}^2 + Ma_{rel}^2)] + \left(-\frac{dP}{P} + \tan\beta d\beta\right) [-(\gamma - 1) Ma_{rel}^2]$$

$$\frac{dT}{T} = \frac{dr}{r} \left[\frac{(\gamma - 1) (Ma_{\Omega}^2 + Ma_{rel}^2)}{1 + (\gamma - 1) Ma_{rel}^2} \right] + \left(-\frac{dP}{P} - \tan\beta d\beta\right) \left[-\frac{(\gamma - 1) Ma_{rel}^2}{1 + (\gamma - 1) Ma_{rel}^2} \right]$$

Substituting above expression in Gibbs equation (eqn.(A.5)):

$$\frac{ds}{c_p} = \frac{dT}{T} - \frac{(\gamma - 1)}{\gamma} \frac{dP}{P}$$

$$\frac{ds}{c_p} = \frac{dr}{r} \left[\frac{(\gamma - 1) (Ma_{\Omega}^2 + Ma_{rel}^2)}{1 + (\gamma - 1) Ma_{rel}^2} \right] + \left(-\frac{dP}{P} - \tan\beta \, d\beta \right) \left[-\frac{(\gamma - 1) Ma_{rel}^2}{1 + (\gamma - 1) Ma_{rel}^2} \right] - \frac{(\gamma - 1)}{\gamma} \frac{dP}{P}$$

Collecting pressure terms:

$$\begin{aligned} \frac{dP}{P} \left[\frac{(\gamma - 1) Ma_{rel}^2}{1 + (\gamma - 1) Ma_{rel}^2} + \frac{(\gamma - 1)}{\gamma} \right] &= \frac{dP}{P} \left[\frac{\gamma (\gamma - 1) Ma_{rel}^2}{\gamma (1 + (\gamma - 1) Ma_{rel}^2)} + \frac{(\gamma - 1) (1 + (\gamma - 1) Ma_{rel}^2)}{\gamma (1 + (\gamma - 1) Ma_{rel}^2)} \right] \\ &= \frac{dP}{P} \left[\frac{(\gamma - 1) (\gamma Ma_{rel}^2 + (1 + (\gamma - 1) Ma_{rel}^2))}{\gamma (1 + (\gamma - 1) Ma_{rel}^2)} \right] = \frac{dP}{P} \left[\frac{(\gamma - 1) (\gamma Ma_{rel}^2 + 1 + \gamma Ma_{rel}^2 - Ma_{rel}^2)}{\gamma (1 + (\gamma - 1) Ma_{rel}^2)} \right] \\ &= \frac{dP}{P} \left[\frac{(\gamma - 1) (2\gamma Ma_{rel}^2 + 1 - Ma_{rel}^2)}{\gamma (1 + (\gamma - 1) Ma_{rel}^2)} \right] \end{aligned}$$

Gibbs equation is finally written as:

$$\begin{aligned} \frac{ds}{c_p} = \frac{dr}{r} \left[\frac{(\gamma - 1) (Ma_{\Omega}^2 + Ma_{rel}^2)}{1 + (\gamma - 1) Ma_{rel}^2} \right] + \frac{dP}{P} \left[\frac{(\gamma - 1) (2\gamma Ma_{rel}^2 + 1 - Ma_{rel}^2)}{\gamma (1 + (\gamma - 1) Ma_{rel}^2)} \right] \\ - \tan\beta \, d\beta \left[\frac{(\gamma - 1) Ma_{rel}^2}{1 + (\gamma - 1) Ma_{rel}^2} \right] \end{aligned}$$

The pressure gradient in the normal to the relative streamline direction the equation assuming zero circumferential gradient is:

$$-\frac{1}{\rho} \frac{\partial P}{\partial n} = \frac{1}{\rho} \frac{\partial P}{\partial m} \sin\beta$$

And the viscous force is approximated using Marble's equation as:

$$f_{p,visc} = T \frac{\partial s}{\partial m} \cos\beta$$

Dividing by m and multiply by $\cos\beta$:

$$\frac{\partial s}{c_p \partial m} \cos\beta = \frac{\partial r}{\partial m} \cos\beta \left[\frac{(\gamma - 1) (Ma_\Omega^2 + Ma_{rel}^2)}{1 + (\gamma - 1) Ma_{rel}^2} \right] + \frac{dP}{\partial m} \cos\beta \left[\frac{(\gamma - 1) (2\gamma Ma_{rel}^2 + 1 - M_{rel}^2)}{\gamma (1 + (\gamma - 1) Ma_{rel}^2)} \right] - \tan\beta \cos\beta \frac{\partial \beta}{\partial m} \left[\frac{(\gamma - 1) Ma_{rel}^2}{1 + (\gamma - 1) Ma_{rel}^2} \right]$$

$$\frac{f_{visc}}{c_p} = \frac{\sin\varphi \cos\beta}{r} \left[\frac{(\gamma - 1) (Ma_\Omega^2 + Ma_{rel}^2)}{1 + (\gamma - 1) Ma_{rel}^2} \right] + \frac{\partial P}{\partial m} \frac{\cos\beta}{P} \left[\frac{(\gamma - 1) ((2\gamma - 1) Ma_{rel}^2 + 1)}{\gamma (1 + (\gamma - 1) Ma_{rel}^2)} \right] - \sin\beta \frac{\partial \beta}{\partial m} \left[\frac{(\gamma - 1) Ma_{rel}^2}{1 + (\gamma - 1) Ma_{rel}^2} \right]$$

Solving for the meridional pressure gradient:

$$\frac{\partial P}{\partial m} = \frac{f_{visc}}{c_p} \frac{P}{\cos\beta} \left[\frac{\gamma (1 + (\gamma - 1) Ma_{rel}^2)}{(\gamma - 1) ((2\gamma - 1) Ma_{rel}^2 + 1)} \right] - \frac{\sin\varphi \cos\beta}{r} \frac{P}{\cos\beta} \left[\frac{\gamma (1 + (\gamma - 1) Ma_{rel}^2)}{(\gamma - 1) ((2\gamma - 1) Ma_{rel}^2 + 1)} \right] \left[\frac{(\gamma - 1) (Ma_\Omega^2 + Ma_{rel}^2)}{1 + (\gamma - 1) Ma_{rel}^2} \right] + P \frac{\sin\beta}{\cos\beta} \frac{\partial \beta}{\partial m} \left[\frac{\gamma (1 + (\gamma - 1) Ma_{rel}^2)}{(\gamma - 1) ((2\gamma - 1) Ma_{rel}^2 + 1)} \right] \left[\frac{(\gamma - 1) Ma_{rel}^2}{1 + (\gamma - 1) Ma_{rel}^2} \right]$$

$$\frac{\partial P}{\partial m} = \frac{f_{visc}}{c_p} \frac{P}{\cos\beta} \left[\frac{\gamma (1 + (\gamma - 1) Ma_{rel}^2)}{(\gamma - 1) ((2\gamma - 1) Ma_{rel}^2 + 1)} \right] - \frac{\sin\varphi \cos\beta}{r} \frac{P}{\cos\beta} \left[\frac{\gamma (Ma_\Omega^2 + Ma_{rel}^2)}{((2\gamma - 1) Ma_{rel}^2 + 1)} \right] + P \frac{\sin\beta}{\cos\beta} \frac{\partial \beta}{\partial m} \left[\frac{\gamma Ma_{rel}^2}{((2\gamma - 1) Ma_{rel}^2 + 1)} \right]$$

Multiplying by $-\sin\beta$ to obtain the normal pressure gradient (note that $\partial/\partial\theta = 0$):

$$\frac{\partial P}{\partial m} = \frac{f_{visc}}{c_p} \frac{P}{\cos\beta} \left[\frac{\gamma (1 + (\gamma - 1) Ma_{rel}^2)}{(\gamma - 1) ((2\gamma - 1) Ma_{rel}^2 + 1)} \right] - \frac{\sin\varphi \cos\beta}{r} \frac{P}{\cos\beta} \left[\frac{\gamma (Ma_\Omega^2 + Ma_{rel}^2)}{((2\gamma - 1) Ma_{rel}^2 + 1)} \right] + P \frac{\sin\beta}{\cos\beta} \frac{\partial \beta}{\partial m} \left[\frac{\gamma Ma_{rel}^2}{((2\gamma - 1) Ma_{rel}^2 + 1)} \right]$$

Multiplying by $-\sin\beta$ to get the normal pressure gradient (since the tangential pressure gradient is zero only the meridional is needed):

$$\frac{\partial P}{\partial n} = -\frac{f_{visc}}{c_p} \tan\beta \, P \left[\frac{\gamma (1 + (\gamma - 1) Ma_{rel}^2)}{(\gamma - 1) ((2\gamma - 1) Ma_{rel}^2 + 1)} \right] + \frac{\sin\varphi \cos\beta}{r} \tan\beta \, P \left[\frac{\gamma (Ma_\Omega^2 + Ma_{rel}^2)}{((2\gamma - 1) Ma_{rel}^2 + 1)} \right] \\ - \sin\beta \tan\beta \, P \frac{\partial\beta}{\partial m} \left[\frac{\gamma Ma_{rel}^2}{((2\gamma - 1) Ma_{rel}^2 + 1)} \right]$$

Rewriting the terms, dividing by density and using the following relations:

$$\frac{P}{\rho} = \frac{u^2}{\gamma Ma^2}$$

$$\frac{u^2}{\gamma RT} = Ma^2$$

$$c_p = \frac{\gamma R}{\gamma - 1}$$

$$\frac{1}{\rho} \frac{\partial P}{\partial n} = -\frac{f_{visc}}{R} \tan\beta \, w^2 \left[\frac{\left(\frac{1}{Ma_{rel}^2} + (\gamma - 1) \right)}{(\gamma - 1) ((2\gamma - 1) Ma_{rel}^2 + 1)} \right] + \frac{\sin\varphi \sin\beta}{r} w^2 \left[\frac{\left(\frac{Ma_\Omega^2}{Ma_{rel}^2} + 1 \right)}{((2\gamma - 1) Ma_{rel}^2 + 1)} \right] \\ - \sin\beta \tan\beta \, \frac{\partial\beta}{\partial m} w^2 \left[\frac{1}{((2\gamma - 1) Ma_{rel}^2 + 1)} \right]$$

$$\frac{1}{\rho} \frac{\partial P}{\partial n} = \frac{w^2}{(2\gamma - 1) Ma_{rel}^2 + 1} \left[-f_{visc} \tan\beta \frac{\left(\frac{1}{Ma_{rel}^2} + (\gamma - 1) \right)}{(\gamma - 1) R} + \frac{\sin\varphi \sin\beta}{r} \left(\frac{Ma_\Omega^2}{Ma_{rel}^2} + 1 \right) \right. \\ \left. - \sin\beta \tan\beta \, \frac{\partial\beta}{\partial m} \right]$$

The body force balance along a streamline in the rotating FoR is given by:

$$w^2 \left(\left(\frac{\partial\beta}{\partial m} \cos\beta \right) + \frac{\sin\varphi \sin\beta}{r} \right) + \frac{1}{\rho} \frac{\partial P}{\partial n} = -[\sin\varphi] 2w\Omega - [\sin\varphi \sin\beta] \Omega^2 r + f_{n,turn}$$

Substituting the pressure gradient term with the above expression gives:

$$\begin{aligned}
& w^2 \left(\left(\frac{\partial \beta}{\partial m} \cos \beta \right) + \frac{\sin \varphi \sin \beta}{r} \right) + \frac{w^2}{(2\gamma - 1) Ma_{rel}^2 + 1} \left[-f_{visc} \tan \beta \frac{\left(\frac{1}{Ma_{rel}^2} + (\gamma - 1) \right)}{(\gamma - 1) R} \right. \\
& \left. + \frac{\sin \varphi \sin \beta}{r} \left(\frac{Ma_{\Omega}^2}{Ma_{rel}^2} + 1 \right) - \sin \beta \tan \beta \frac{\partial \beta}{\partial m} \right] = -[\sin \varphi] 2w\Omega - [\sin \varphi \sin \beta] \Omega^2 r + f_{n,turn}
\end{aligned}$$

The final expression of the turning force model is finally given by eqn.(A.7):

$$\begin{aligned}
f_{n,turn} = & w^2 \left[\frac{\cos \beta \left((2\gamma - 1) Ma_{rel}^2 - (\tan \beta)^2 + 1 \right)}{(2\gamma - 1) Ma_{rel}^2 + 1} \frac{\partial \beta}{\partial m} + \left(\frac{\frac{Ma_{\Omega}^2}{Ma_{rel}^2} + (2\gamma - 1) Ma_{rel}^2 + 2}{(2\gamma - 1) Ma_{rel}^2 + 1} \right) \right. \\
& \left. \frac{\sin \varphi \sin \beta}{r} \right] + f_{visc} \left[- \frac{\tan \beta \left(\frac{1}{Ma_{rel}^2} + (\gamma - 1) \right)}{(\gamma - 1) R} \right] + 2w\Omega [\sin \varphi] + \Omega^2 r [\sin \varphi \sin \beta]
\end{aligned}
\tag{A.7}$$

Appendix B

Viscous force model derivation

The Gibbs equation can be expressed using stagnation properties [126]. Using stagnation properties in the relative FoR gives::

$$\begin{aligned}
 T_{0,rel} ds &= dh_{0,rel} - \frac{dP_{0,rel}}{\rho_{0,rel}} \\
 T \left(1 + \frac{\gamma - 1}{2} M \alpha_{rel}^2 \right) ds &= dh_{0,rel} - \frac{dP_{0,rel}}{\rho_{0,rel}} \\
 T ds &= - \frac{dh_{0,rel} - \frac{dP_{0,rel}}{\rho_{0,rel}}}{\left(1 + \frac{\gamma - 1}{2} M \alpha_{rel}^2 \right)} \\
 \frac{T ds}{dm} \cos \beta &= - \frac{dh_{0,rel} - \frac{dP_{0,rel}}{\rho_{0,rel}}}{\left(1 + \frac{\gamma - 1}{2} M \alpha_{rel}^2 \right)} \frac{\cos \beta}{dm}
 \end{aligned} \tag{B.1}$$

Rothalpy using the relative stagnation enthalpy is defined as:

$$I = h_{0,rel} - \frac{1}{2} \Omega^2 r^2$$

Differentiating gives:

$$dI = dh_{0,rel} - d \left(\frac{1}{2} \Omega^2 r^2 \right) = dh_{0,rel} - \left(r^2 \Omega d\Omega + \Omega^2 r dr \right)$$

Rothalpy is conserved along a relative streamline, $dI = 0$ and the rotational speed is constant $d\Omega = 0$

$$0 = dh_{0,rel} - \Omega^2 r dr$$

$$\Omega^2 r dr = dh_{0,rel} \quad (B.2)$$

Combining eqns.(B.3) and (B.2):

$$\begin{aligned} \frac{Tds}{dm} \cos\beta &= \frac{\cos\beta}{\left(1 + \frac{\gamma-1}{2} M\alpha_{rel}^2\right)} \left\{ \frac{dh_{0,rel}}{dm} - \frac{1}{\rho_{0,rel}} \frac{dP_{0,rel}}{dm} \right\} \\ \frac{Tds}{dm} \cos\beta &= \frac{\cos\beta}{\left(1 + \frac{\gamma-1}{2} M\alpha_{rel}^2\right)} \left\{ \Omega^2 r \frac{dr}{dm} - \frac{1}{\rho_{0,rel}} \frac{dP_{0,rel}}{dm} \right\} \end{aligned}$$

Using the flow path angle definition:

$$\frac{dr}{dm} = \sin\varphi$$

Substituting the above expression and collecting terms gives:

$$\frac{Tds}{dm} \cos\beta = \frac{\cos\beta \sin\varphi}{\left(1 + \frac{\gamma-1}{2} M\alpha_{rel}^2\right)} \Omega^2 r - \frac{\cos\beta}{\rho_{0,rel} \left(1 + \frac{\gamma-1}{2} M\alpha_{rel}^2\right)} \frac{dP_{0,rel}}{dm}$$

Expressing stagnation density in terms of static density and relative Mach number:

$$\rho_{0,rel} = \rho \left(1 + \frac{\gamma-1}{2} M\alpha_{rel}^2\right)^{\frac{1}{\gamma-1}}$$

Substituting the above expression gives:

$$\frac{Tds}{dm} \cos\beta = \frac{\cos\beta \sin\varphi}{\left(1 + \frac{\gamma-1}{2} M\alpha_{rel}^2\right)} \Omega^2 r - \frac{\cos\beta}{\rho \left(1 + \frac{\gamma-1}{2} M\alpha_{rel}^2\right)^{\frac{1}{\gamma-1}} \left(1 + \frac{\gamma-1}{2} M\alpha_{rel}^2\right)} \frac{dP_{0,rel}}{dm}$$

$$\frac{Tds}{dm} \cos\beta = \frac{\cos\beta \sin\varphi}{\left(1 + \frac{\gamma-1}{2} M\alpha_{rel}^2\right)} \Omega^2 r - \frac{\cos\beta}{\rho \left(1 + \frac{\gamma-1}{2} M\alpha_{rel}^2\right)^{\frac{\gamma}{\gamma-1}}} \frac{dP_{0,rel}}{dm}$$

$$\frac{Tds}{dm} \cos\beta = \frac{\cos\beta \sin\varphi}{\left(1 + \frac{\gamma-1}{2} M \alpha_{rel}^2\right)} \frac{\Omega^2 r^2}{r} - \frac{\cos\beta}{\rho \left(1 + \frac{\gamma-1}{2} M \alpha_{rel}^2\right)^{\frac{\gamma}{\gamma-1}}} \frac{dP_{0,rel}}{dm}$$

Expressing Ω^2 in terms of Mach number:

$$Ma = \frac{V}{\sqrt{\frac{\gamma P}{\rho}}}$$

$$\Omega^2 r^2 = \frac{\gamma P}{\rho} Ma_{\Omega}^2$$

Substitution of the above expression and collecting terms gives the final viscous force expression:

$$F_{visc} = -\frac{Tds}{dm} \cos\beta = \cos\beta \frac{P}{\rho} \left[\frac{1}{P_{0,rel}} \frac{dP_{0,rel}}{dm} - \frac{\sin\varphi}{\left(1 + \frac{\gamma-1}{2} M \alpha_{rel}^2\right)} \frac{\gamma}{r} \frac{Ma_{\Omega}^2}{r} \right] \quad (B.3)$$

Appendix C

Derivation of blockage-mixing model

Using a control volume analysis, assuming incompressible, fully reverse flow, no deviation at the LE and denoting properties at the TE and the LE with subscripts 1 and 2 respectively, mass flow, momentum and energy conservation are combined to derive the blockage-mixing model (see section(3.5))

Mass flow conservation:

$$\dot{m} = -\rho W_1 \cos \beta_1 A_1 = -\rho W_2 \cos \kappa_2 A_2 \overline{b_{sep}}$$

$$W_2 = \frac{W_1 \cos \beta_1 A_1}{\cos \kappa_2 A_2 \overline{b_{sep}}} \quad (C.1)$$

,where at the outlet a zero degree deviation angle was assumed.

Rotary stagnation pressure conservation inside the isentropic flow region:

$$\begin{aligned} P_{\omega 2,i} &= P_{\omega 1} \\ P_2 + \frac{1}{2}\rho (W_2^2 - U_2^2) &= P_1 + \frac{1}{2}\rho (W_1^2 - U_1^2) \\ P_2 &= P_1 + \frac{1}{2}\rho (W_1^2 - U_1^2) - \frac{1}{2}\rho (W_2^2 - U_2^2) \\ P_2 &= P_1 + \frac{1}{2}\rho (W_1^2 - W_2^2) - \frac{1}{2}\rho (U_1^2 - U_2^2) \end{aligned} \quad (C.2)$$

Momentum conservation along the full duct and parallel to the streamwise coordinate:

$$P_2 A_2 \cos \kappa_2 - P_1 A_1 \cos \kappa_1 + \left(\rho W_2^2 A_2 \overline{b_{sep}} \cos(\kappa_2) \right) - \left(\rho W_1^2 \cos(\beta_1 - \kappa_1) \cos \beta_1 A_1 \right) = \iiint \rho f_{cyl} \hat{l} dV$$

Collecting terms:

$$P_2 - P_1 \frac{A_1 \cos \kappa_1}{A_2 \cos \kappa_2} + \left(\rho W_2^2 \overline{b_{sep}} \right) - \left(\rho W_1^2 \frac{A_1 \cos \beta_1 \cos(\beta_1 - \kappa_1)}{A_2 \cos \kappa_2} \right) = \frac{1}{A_2 \cos \kappa_2} \iiint \rho f_{cyl} \hat{l} dV$$

Solving the above equation for P_2 and substituting it in the energy equation (eqn.(C.2)) gives:

$$P_1 + \frac{1}{2} \rho (W_1^2 - W_2^2) - \frac{1}{2} \rho (U_1^2 - U_2^2) - P_1 \frac{A_1 \cos \kappa_1}{A_2 \cos \kappa_2} + \left(\rho W_2^2 \overline{b_{sep}} \right) - \left(\rho W_1^2 \frac{A_1 \cos \beta_1 \cos(\beta_1 - \kappa_1)}{A_2 \cos \kappa_2} \right) = \frac{1}{A_2 \cos \kappa_2} \iiint \rho f_{cyl} \hat{l} dV$$

$$\frac{1}{2} \rho W_2^2 (2 \overline{b_{sep}} - 1) = -P_1 - \frac{1}{2} \rho W_1^2 - \frac{1}{2} \rho (U_2^2 - U_1^2) + P_1 \frac{A_1 \cos \kappa_1}{A_2 \cos \kappa_2} + \left(\rho W_1^2 \frac{A_1 \cos \beta_1 \cos(\beta_1 - \kappa_1)}{A_2 \cos \kappa_2} \right) + \frac{1}{A_2 \cos \kappa_2} \iiint \rho f_{cyl} \hat{l} dV$$

Substituting continuity (eqn.(C.2)) in the above equation gives:

$$\frac{1}{2} \rho W_1^2 \left(\frac{\cos \beta_1 A_1}{\cos \kappa_2 A_2} \right)^2 \frac{(2 \overline{b_{sep}} - 1)}{\overline{b_{sep}}^2} = -P_1 - \frac{1}{2} \rho W_1^2 - \frac{1}{2} \rho (U_2^2 - U_1^2) + P_1 \frac{A_1 \cos \kappa_1}{A_2 \cos \kappa_2} + \left(\rho W_1^2 \frac{A_1 \cos \beta_1 \cos(\beta_1 - \kappa_1)}{A_2 \cos \kappa_2} \right) + \frac{1}{A_2 \cos \kappa_2} \iiint \rho f_{cyl} \hat{l} dV$$

$$W_1^2 \left(\frac{\cos \beta_1 A_1}{\cos \kappa_2 A_2} \right)^2 \frac{(2 \overline{b_{sep}} - 1)}{\overline{b_{sep}}^2} = -2 \frac{P_1}{\rho} - W_1^2 - (U_2^2 - U_1^2) + 2 \frac{P_1 A_1 \cos \kappa_1}{\rho A_2 \cos \kappa_2} + 2 \left(W_1^2 \frac{A_1 \cos \beta_1 \cos(\beta_1 - \kappa_1)}{A_2 \cos \kappa_2} \right) + \frac{2}{A_2 \cos \kappa_2} \iiint f_{cyl} \hat{l} dV$$

$$\begin{aligned} \frac{(2\overline{b_{sep}} - 1)}{\overline{b_{sep}}^2} = & \left(\frac{\cos\kappa_2 A_2}{\cos\beta_1 A_1} \right)^2 \left[-2 \frac{P_1}{\rho W_1^2} - 1 - \frac{(U_2^2 - U_1^2)}{W_1^2} + 2 \frac{P_1}{\rho W_1^2} \frac{A_1 \cos\kappa_1}{A_2 \cos\kappa_2} \right. \\ & \left. + 2 \left(\frac{A_1 \cos\beta_1 \cos(\beta_1 - \kappa_1)}{A_2 \cos\kappa_2} \right) + \frac{2}{A_2 \cos\kappa_2 W_1^2} \iiint f_{cyl} \hat{l} dV \right] \end{aligned}$$

$$\boxed{\overline{b_{sep}}^{-2} \left\{ \left(\frac{\cos\kappa_2 A_2}{\cos\beta_1 A_1} \right)^2 \left[2 \frac{P_1}{\rho W_1^2} \left(\frac{A_1 \cos\kappa_1}{A_2 \cos\kappa_2} - 1 \right) - \frac{(U_2^2 - U_1^2)}{W_1^2} + 2 \left(\frac{A_1 \cos\beta_1 \cos(\beta_1 - \kappa_1)}{A_2 \cos\kappa_2} \right) \right. \right.} \\ \left. \left. + \frac{2}{A_2 \cos\kappa_2 W_1^2} \iiint f_{cyl} \hat{l} dV - 1 \right] \right\} - 2\overline{b_{sep}} + 1 = 0} \quad (C.3)$$

The above equation is a quadratic function and can be written using coefficients as follows:

$$a \overline{b_{sep}}^2 - 2\overline{b_{sep}} + 1 = 0$$

The blockage parameter is obtained by solving the quadratic function as follows:

$$\boxed{\overline{b_{sep}} = \frac{2 \pm \sqrt{4(1 - \alpha)}}{2a}} \quad (C.4)$$

In the above equations, the projection in the streamwise direction of the source terms due to the formulation of the equations in the cylindrical coordinate system, is given by:

$$\iiint \rho f_{cyl} \hat{l} dV = \iiint \frac{\rho}{r} \sin\varphi \left\{ \cos|\kappa_{bl}| \left[(W_\theta + \omega r)^2 \right] - W_m \sin|\kappa_{bl}| (W_\theta + 2\omega r) \right\} \hat{l} dV$$

The cylindrical source term can be evaluated by setting

$$\beta_1 = \kappa_1$$

$$b = 1$$

This essentially implies that at a zero incidence angle the blockage parameter should be 1 (no aerodynamic blockage).

Setting the flow angle at the TE equal to κ_1 and the blockage parameter equal to 1, the quadratic function can be solved for the body force source terms as follows:

$$\left(\frac{\cos\kappa_2 A_2}{\cos\beta_1 A_1}\right)^2 \left[2\frac{P_1}{\rho W_1^2} \left(\frac{A_1 \cos\kappa_1}{A_2 \cos\kappa_2} - 1\right) - \frac{(U_2^2 - U_1^2)}{W_1^2} + 2\left(\frac{A_1 \cos\beta_1 \cos(\beta_1 - \kappa_1)}{A_2 \cos\kappa_2}\right) + \frac{2}{A_2 \cos\kappa_2 W_1^2} \iiint f_{cyl} \hat{l} dV - 1 \right] - 2 + 1 = 0$$

$$2\frac{P_1}{\rho W_1^2} \left(\frac{A_1 \cos\kappa_1}{A_2 \cos\kappa_2} - 1\right) - \frac{(U_2^2 - U_1^2)}{W_1^2} + 2\left(\frac{A_1 \cos\kappa_1}{A_2 \cos\kappa_2}\right) + \frac{2}{A_2 \cos\kappa_2 W_1^2} \iiint f_{cyl} \hat{l} dV - 1 = \left(\frac{\cos\kappa_1 A_1}{\cos\kappa_2 A_2}\right)^2$$

$$\iiint f_{cyl} \hat{l} dV = \frac{A_2 \cos\kappa_2 W_1^2}{2} \left\{ \left(\frac{\cos\kappa_1 A_1}{\cos\kappa_2 A_2}\right)^2 - 2\frac{P_1}{\rho W_1^2} \left(\frac{\cos\kappa_1 A_1}{\cos\kappa_2 A_2} - 1\right) + \frac{(U_2^2 - U_1^2)}{W_1^2} - 2\left(\frac{\cos\kappa_1 A_1}{\cos\kappa_2 A_2}\right) + 1 \right\}$$

The body force source terms are defined analytically using eqn.(C.5):

$$\boxed{\iiint f_{cyl} \hat{l} dV = \frac{A_2 \cos\kappa_2 W_1^2}{2} \left\{ \left(\left(\frac{\cos\kappa_1 A_1}{\cos\kappa_2 A_2}\right) - 1\right)^2 - 2\frac{P_1}{\rho W_1^2} \left(\frac{A_1 \cos\kappa_1}{A_2 \cos\kappa_2} - 1\right) + \frac{(U_2^2 - U_1^2)}{W_1^2} \right\}} \quad (C.5)$$

Constant area mixing - Axial duct

Assuming constant area mixing, incompressible flow, no deviation at the LE and denoting with subscript 2 and 3 properties at the blade LE and the downstream boundary of the control volume respectively (see figure(3.14)), continuity, circumferential and axial momentum can be written as follows:

Continuity:

$$\rho V_3 A_3 \cos\beta_3 = \rho V_{2w} A_2 (1 - \overline{b_{sep}}) \cos\beta_2 + \rho V_{2j} A_2 \overline{b_{sep}} \cos\beta_2$$

$$V_3 \cos\beta_3 = V_{2w} (1 - \overline{b_{sep}}) \cos\kappa_2 + V_{2j} \overline{b_{sep}} \cos\kappa_2$$

Circumferential momentum:

$$\rho V_{3x} V_{3\theta} A_3 = \rho V_{2jx} V_{2j\theta} A_2 \overline{b_{sep}} + \rho V_{2wx} V_{2w\theta} A_2 (1 - \overline{b_{sep}})$$

$$\rho \sin \beta_3 \cos \beta_3 V_3^2 A_3 = \rho \sin \beta_{2j} \cos \beta_{2j} V_{2j}^2 A_2 \overline{b_{sep}} + \rho \sin \beta_{2w} \cos \beta_{2w} V_{2w}^2 A_2 (1 - \overline{b_{sep}})$$

Assuming constant area mixing and that both streams leave the blade with zero deviation angle:

$$\sin 2\beta_3 V_3^2 = \sin 2\kappa_{2j} V_{2j}^2 \overline{b_{sep}} + \sin 2\kappa_{2w} V_{2w}^2 (1 - \overline{b_{sep}})$$

Or

$$\sin 2\beta_3 V_3^2 = \sin 2\kappa_2 V_{2j}^2 \overline{b_{sep}} + \sin 2\kappa_2 V_{2w}^2 (1 - \overline{b_{sep}})$$

$$\sin 2\beta_3 V_3^2 = \sin 2\kappa_2 (V_{2j}^2 \overline{b_{sep}} + V_{2w}^2 (1 - \overline{b_{sep}}))$$

Axial momentum:

$$\rho \cos \beta_3^2 V_3^2 A_3 + P_3 A_3 = \rho \cos \beta_{2j}^2 V_{2j}^2 A_2 \overline{b_{sep}} + \rho \cos \beta_{2w}^2 V_{2w}^2 A_2 (1 - \overline{b_{sep}}) + P_2 A_2$$

Assuming constant area mixing and that both streams leave the blade with zero deviation angle:

$$\cos \beta_3^2 V_3^2 + \frac{P_3}{\rho} = \cos \kappa_2^2 V_{2j}^2 \overline{b_{sep}} + \cos \kappa_{2w}^2 V_{2w}^2 (1 - \overline{b_{sep}}) + \frac{P_2}{\rho}$$

Using continuity and assuming zero wake velocity:

$$V_3 \cos \beta_3 = V_{2w} (1 - \overline{b_{sep}}) \cos \kappa_2 + V_{2j} \overline{b_{sep}} \cos \kappa_2$$

$$V_3 = V_{2j} \overline{b_{sep}} \frac{\cos \kappa_2}{\cos \beta_3}$$

Using circumferential momentum and assuming zero wake velocity:

$$\sin 2\beta_3 V_3^2 = \sin 2\kappa_2 (V_{2j}^2 \overline{b_{sep}} + V_{2w}^2 (1 - \overline{b_{sep}}))$$

$$\cos \beta_3 \sin \beta_3 V_3^2 = \frac{\sin 2\kappa_2}{2} V_{2j}^2 \overline{b_{sep}}$$

$$\cos \beta_3 \sin \beta_3 \left(V_{2j} \overline{b_{sep}} \frac{\cos \kappa_2}{\cos \beta_3} \right)^2 = \sin \kappa_2 \cos \kappa_2 V_{2j}^2 \overline{b_{sep}}$$

$$\cos \beta_3 \sin \beta_3 V_{2j}^2 \overline{b_{sep}}^2 \left(\frac{\cos \kappa_2}{\cos \beta_3} \right)^2 = \sin \kappa_2 \cos \kappa_2 V_{2j}^2 \overline{b_{sep}}$$

$$\tan\beta_3 \overline{b_{sep}} \cos\kappa_2 = \sin\kappa_2$$

The mixed-out flow angle is given by:

$$\boxed{\tan\beta_3 = \frac{\tan\kappa_2}{\overline{b_{sep}}}} \quad (C.6)$$

The deviation angle due to mixing-out of the flow is given by:

$$\boxed{\delta_3 = \beta_3 - \kappa_2 = \operatorname{atan}\left(\frac{\tan\kappa_2}{\overline{b_{sep}}}\right) - \kappa_2} \quad (C.7)$$

Now V_3 can be evaluated using continuity:

$$V_3 = V_{2j} \overline{b_{sep}} \frac{\cos\kappa_2}{\cos\beta_3}$$

Using axial momentum and assuming zero wake velocity:

$$\begin{aligned} \cos\beta_3^2 V_3^2 + \frac{P_3}{\rho} &= \cos\kappa_{2j}^2 V_{2j}^2 \overline{b_{sep}} + \rho \cos\kappa_{2w}^2 V_{2w}^2 (1 - \overline{b_{sep}}) + \frac{P_2}{\rho} \\ \rho \cos\beta_3^2 V_3^2 + P_3 &= \rho \cos\kappa_{2j}^2 V_{2j}^2 \overline{b_{sep}} + P_2 \end{aligned}$$

Add and subtract the dynamic heads to get stagnation pressure:

$$\begin{aligned} \rho \cos\beta_3^2 V_3^2 - 0.5\rho V_3^2 + P_{03} &= \rho \cos\kappa_{2j}^2 V_{2j}^2 \overline{b_{sep}} - 0.5\rho V_{2j}^2 + P_{02} \\ \rho V_{2j}^2 \overline{b_{sep}}^2 \cos\kappa_2^2 - \frac{0.5\rho V_{2j}^2 \overline{b_{sep}}^2 \cos\kappa_2^2}{\cos\beta_3^2} + P_{03} &= \rho \cos\kappa_{2j}^2 V_{2j}^2 \overline{b_{sep}} - 0.5\rho V_{2j}^2 + P_{02} \\ 2\overline{b_{sep}}^2 \cos\kappa_2^2 - \frac{\overline{b_{sep}}^2 \cos\kappa_2^2}{\cos\beta_3^2} + \frac{P_{03}}{0.5\rho V_{2j}^2} &= 2\cos\kappa_{2j}^2 \overline{b_{sep}} - 1 + \frac{P_{02}}{0.5\rho V_{2j}^2} \\ \frac{P_{02} - P_{03}}{0.5\rho V_{2j}^2} &= 2\overline{b_{sep}}^2 \cos\kappa_2^2 - \frac{\overline{b_{sep}}^2 \cos\kappa_2^2}{\cos\beta_3^2} - 2\cos\kappa_{2j}^2 \overline{b_{sep}} + 1 \end{aligned}$$

The loss coefficient due to mixing-out of the jet flow and the wake is finally given by:

$$\boxed{\frac{P_{02} - P_{03}}{0.5\rho V_{2j}^2} = \cos\kappa_2^2 \left(\overline{b_{sep}}^2 \left(2 - \frac{1}{\cos\beta_3^2} \right) - 2 \overline{b_{sep}} + 1/\cos\kappa_2^2 \right)} \quad (C.8)$$

Variable area mixing - Radial duct

Assuming strictly radial flow, variable area mixing, incompressible flow, no deviation at the LE, no angular velocity (stator) and denoting with subscript 2 and 3 properties at the blade LE and the downstream boundary of the control volume respectively (see figure(3.15)), continuity, circumferential and axial momentum can be written as follows:

Continuity:

$$\rho V_3 A_3 \cos \beta_3 = \rho V_{2j} A_2 \overline{b_{sep}} \cos \kappa_2$$

$$V_3 = \frac{V_{2j} A_2 \overline{b_{sep}} \cos \kappa_2}{A_3 \cos \beta_3}$$

Circumferential momentum:

$$\rho V_{3m} V_{3\theta} A_3 = \rho V_{2jm} V_{2j\theta} A_2 \overline{b_{sep}} - \iiint \frac{w_m \sin \varphi}{r} [w_\theta + 2\Omega r] dV$$

The mixing is assumed to happen outside the blade, $\Omega=0$ and $w_\theta = V_\theta$.

$$\rho \sin \beta_3 \cos \beta_3 V_3^2 A_3 = \rho \sin \beta_{2j} \cos \beta_{2j} V_{2j}^2 A_2 \overline{b_{sep}} - \iiint \frac{V^2 \cos \beta \sin \beta \sin \varphi}{r} dV$$

Assuming no deviation at the LE:

$$\rho \sin \beta_3 \cos \beta_3 V_3^2 A_3 = \rho \sin \kappa_{2j} \cos \kappa_2 V_{2j}^2 A_2 \overline{b_{sep}} - \iiint \rho \frac{V^2 \cos \beta \sin \beta \sin \varphi}{r} dV$$

The integration part can either be directly evaluated (numerically) using the solver data or simplified as follows:

$$\iiint \rho \frac{V^2 \cos \beta \sin \beta \sin \varphi}{r} dV$$

The velocity could be taken in the middle of the volume and the velocity can be computed using continuity:

$$\rho V_{2j} A_2 \overline{b_{sep}} \cos \kappa_2 = \rho V_{23} \frac{(A_2 + A_3)}{2} \cos \kappa_2$$

$$V_{23} = \frac{2 V_{2j} A_2 \overline{b_{sep}}}{(A_2 + A_3)}$$

Assuming a purely radial duct, the control volume can be estimated as:

$$Vol = \frac{2\pi (r_2 + r_3)}{2} (r_2 - r_3) \frac{A_2}{2\pi r_2} = \frac{(r_2^2 - r_3^2)}{r_2} \frac{A_2}{2}$$

$$\iiint \rho \frac{V^2 \cos\beta \sin\beta \sin\varphi}{r} dV \approx \rho \frac{V_{23}^2 \cos\kappa_2 \sin\kappa_2}{(r_2 + r_3)/2} \frac{(r_2^2 - r_3^2)}{r_2} \frac{A_2}{2}$$

$$\iiint \rho \frac{V^2 \cos\beta \sin\beta \sin\varphi}{r} dV \approx \rho \frac{(2 V_{2j} A_2 \overline{b_{sep}})^2}{(A_2 + A_3)^2} \frac{(r_2 - r_3)}{r_2} A_2 \cos\kappa_2 \sin\kappa_2$$

The angular momentum equation can be written now as:

$$\rho \sin\beta_3 \cos\beta_3 V_3^2 A_3 = \rho \sin\kappa_{2j} \cos\kappa_2 V_{2j}^2 A_2 \overline{b_{sep}} - \rho \frac{(2 V_{2j} A_2 \overline{b_{sep}})^2}{(A_2 + A_3)^2} \frac{(r_2 - r_3)}{r_2} A_2 \cos\kappa_2 \sin\kappa_2$$

Collecting terms and replacing V_3^2 using continuity gives:

$$\sin\beta_3 \cos\beta_3 \left(\frac{V_{2j} A_2 \overline{b_{sep}} \cos\kappa_2}{A_3 \cos\beta_3} \right)^2 A_3 = \sin\kappa_2 \cos\kappa_2 A_2 V_{2j}^2 \left(\overline{b_{sep}} - \frac{(2 A_2 \overline{b_{sep}})^2}{(A_2 + A_3)^2} \frac{(r_2 - r_3)}{r_2} \right)$$

$$\tan\beta_3 \cos\kappa_2^2 V_{2j}^2 \frac{\overline{b_{sep}}^2 A_2^2}{A_3^2} = \sin\kappa_2 \cos\kappa_2 V_{2j}^2 \frac{A_2}{A_3} \left(\overline{b_{sep}} - \frac{(2 A_2 \overline{b_{sep}})^2}{(A_2 + A_3)^2} \frac{(r_2 - r_3)}{r_2} \right)$$

$$\tan\beta_3 \overline{b_{sep}}^2 = \tan\kappa_2 \frac{A_2}{A_3} \left(\overline{b_{sep}} - \frac{(2 A_2 \overline{b_{sep}})^2}{(A_2 + A_3)^2} \frac{(r_2 - r_3)}{r_2} \right)$$

$$\tan\beta_3 = \tan\kappa_2 \frac{A_3}{A_2} \left(\frac{1}{\overline{b_{sep}}} - \frac{4A_2^2}{(A_2 + A_3)^2} \frac{(r_2 - r_3)}{r_2} \right)$$

The mixed-out flow angle expressed in terms of areas is given by:

$$\boxed{\tan\beta_3 = \tan\kappa_2 \frac{A_3}{A_2} \left(\frac{1}{\overline{b_{sep}}} - \frac{4A_2^2}{(A_2 + A_3)^2} \frac{(A_2 - A_3)}{A_2} \right)} \quad (C.9)$$

The mixed-out flow angle expressed in terms of radii is given by:

$$\boxed{\tan\beta_3 = \tan\kappa_2 \frac{r_3}{r_2} \left(\frac{1}{\overline{b_{sep}}} - \frac{4}{\left(1 + \frac{r_3}{r_2}\right)^2} \left(1 - \frac{r_3}{r_2}\right) \right)} \quad (C.10)$$

Due to change in radius, the angular velocity changes, meaning that the flow angle will change even when no mixing is taking place. This can be approximated by simply setting the blockage parameter to zero:

$$\beta_{3,i} = \text{atan} \left(\tan\kappa_2 \frac{r_3}{r_2} \left(1 - \frac{4}{\left(1 + \frac{r_3}{r_2}\right)^2} \left(1 - \frac{r_3}{r_2}\right) \right) \right) \quad (C.11)$$

The imposed deviation angle is the difference between the ideal and mixed out relative flow angles:

$$\boxed{\delta_3 = \beta_3 - \beta_{3,i}} \quad (C.12)$$

Meridional momentum:

$$\rho \cos\beta_3^2 V_3^2 A_3 + P_3 A_3 = \rho \cos\kappa_2^2 V_{2j}^2 A_2 \overline{b_{sep}} + P_2 A_2 + \iiint \rho \frac{\sin\varphi}{r} \sin\beta^2 V^2 dV$$

The integral can be simplified by assuming radial-only flow, flow angle equal to blade metal angle at the LE and average velocity as computed earlier:

$$\begin{aligned} \iiint \rho \frac{\sin\varphi}{r} \sin\beta^2 V^2 dV &\approx \rho \frac{\sin\kappa_2^2 V_{23}^2}{(r_2 + r_3)/2} \frac{(r_2^2 - r_3^2)}{r_2} \frac{A_2}{2} \\ \iiint \rho \frac{\sin\varphi}{r} \sin\beta^2 V^2 dV &\approx \rho \sin\kappa_2^2 V_{23}^2 \frac{(r_2 - r_3)}{r_2} A_2 \end{aligned}$$

$$\iiint \rho \frac{\sin\varphi}{r} \sin\beta^2 V^2 dV \approx \rho \sin\kappa_2^2 \left(\frac{2 V_{2j} A_2 \overline{b_{sep}}}{(A_2 + A_3)} \right)^2 \frac{(r_2 - r_3)}{r_2} A_2$$

And the meridional momentum equation can be written as:

$$\rho \cos\beta_3^2 V_3^2 A_3 + P_3 A_3 = \rho \cos\kappa_2^2 V_{2j}^2 A_2 \overline{b_{sep}} + P_2 A_2 + \iiint \rho \frac{\sin\varphi}{r} \sin\beta^2 V dV$$

Solving for P_3 :

$$P_3 = \rho \cos\kappa_2^2 V_{2j}^2 \frac{A_2}{A_3} \overline{b_{sep}} - \rho \cos\beta_3^2 V_3^2 + P_2 \frac{A_2}{A_3} + \rho \sin\kappa_2^2 \left(\frac{2 V_{2j} A_2 \overline{b_{sep}}}{(A_2 + A_3)} \right)^2 \frac{(r_2 - r_3)}{r_2} \frac{A_2}{A_3}$$

$$P_3 = \rho \cos \kappa_2^2 V_{2j}^2 \frac{A_2}{A_3} \overline{b_{sep}} - \rho \cos \beta_3^2 \left(\frac{V_{2j} A_2 \overline{b_{sep}} \cos \kappa_2}{A_3 \cos \beta_3} \right)^2 + P_2 \frac{A_2}{A_3} + \rho \sin \kappa_2^2 \left(\frac{2 V_{2j} A_2 \overline{b_{sep}}}{(A_2 + A_3)} \right)^2 \frac{(r_2 - r_3)}{r_2} \frac{A_2}{A_3}$$

$$P_3 = \rho V_{2j}^2 \frac{A_2}{A_3} \left(\cos \kappa_2^2 \overline{b_{sep}} - \frac{A_2}{A_3} \overline{b_{sep}}^2 \cos \kappa_2^2 + 4 \sin \kappa_2^2 \overline{b_{sep}}^2 \left(\frac{r_2}{(r_2 + r_3)} \right)^2 \frac{(r_2 - r_3)}{r_2} \right) + P_2 \frac{A_2}{A_3}$$

The areas can be expressed purely in terms of radii as:

$$P_3 - P_2 \frac{r_2}{r_3} = \rho V_{2j}^2 \frac{r_2}{r_3} \left(\cos \kappa_{2j}^2 \overline{b_{sep}} - \frac{r_2}{r_3} \overline{b_{sep}}^2 \cos \kappa_2^2 + 4 \sin \kappa_2^2 \overline{b_{sep}}^2 \left(\frac{r_2}{(r_2 + r_3)} \right)^2 \frac{(r_2 - r_3)}{r_2} \right)$$

Rewriting the LHS:

$$\begin{aligned} & P_3 - P_2 + P_2 - P_2 \frac{r_2}{r_3} + \frac{1}{2} \rho V_3^2 - \frac{1}{2} \rho V_3^2 + \frac{1}{2} \rho V_{2j}^2 - \frac{1}{2} \rho V_{2j}^2 \\ & P_{03} - P_{02} + P_2 \left(1 - \frac{r_2}{r_3} \right) - \frac{1}{2} \rho \left(\frac{V_{2j} A_2 \overline{b_{sep}} \cos \kappa_2}{A_3 \cos \beta_3} \right)^2 + \frac{1}{2} \rho V_{2j}^2 \\ & - \Delta P_0 + P_2 \left(1 - \frac{r_2}{r_3} \right) - \frac{1}{2} \rho V_{2j}^2 \left(\left(\frac{A_2 \overline{b_{sep}} \cos \kappa_2}{A_3 \cos \beta_3} \right)^2 - 1 \right) \end{aligned}$$

The full equation can now be written as follows:

$$\begin{aligned} -\Delta P_0 + P_2 \left(1 - \frac{r_2}{r_3} \right) - \frac{1}{2} \rho V_{2j}^2 \left(\left(\frac{r_2 \overline{b_{sep}} \cos \kappa_2}{r_3 \cos \beta_3} \right)^2 - 1 \right) &= \rho V_{2j}^2 \frac{r_2}{r_3} \left(\cos \kappa_{2j}^2 \overline{b_{sep}} - \frac{r_2}{r_3} \overline{b_{sep}}^2 \cos \kappa_2^2 \right. \\ &\quad \left. + 4 \sin \kappa_2^2 \overline{b_{sep}}^2 \left(\frac{r_2}{(r_2 + r_3)} \right)^2 \frac{(r_2 - r_3)}{r_2} \right) \end{aligned}$$

$$\begin{aligned} -\frac{\Delta P_0}{\frac{1}{2} \rho V_{2j}^2} &= -\frac{P_2 \left(1 - \frac{r_2}{r_3} \right)}{\frac{1}{2} \rho V_{2j}^2} + \left(\left(\frac{r_2 \overline{b_{sep}} \cos \kappa_2}{r_3 \cos \beta_3} \right)^2 - 1 \right) + 2 \frac{r_2}{r_3} \left(\cos \kappa_{2j}^2 \overline{b_{sep}} - \frac{r_2}{r_3} \overline{b_{sep}}^2 \cos \kappa_2^2 \right. \\ &\quad \left. + 4 \sin \kappa_2^2 \overline{b_{sep}}^2 \left(\frac{r_2}{(r_2 + r_3)} \right)^2 \frac{(r_2 - r_3)}{r_2} \right) \end{aligned}$$

$$\begin{aligned}
-\frac{\Delta P_0}{\frac{1}{2}\rho V_{2j}^2} = & -\frac{P_2 \left(1 - \frac{r_2}{r_3}\right)}{\frac{1}{2}\rho V_{2j}^2} + \left(\left(\frac{r_2 \overline{b_{sep}} \cos \kappa_2}{r_3 \cos \beta_3} \right)^2 - 1 \right) + 2 \frac{r_2}{r_3} \left(\cos^2 \kappa_2 \overline{b_{sep}} - \frac{r_2}{r_3} \overline{b_{sep}}^2 \cos^2 \kappa_2 \right. \\
& \left. + 4 \sin \kappa_2 \overline{b_{sep}}^2 \left(\frac{r_2}{(r_2 + r_3)} \right)^2 \frac{(r_2 - r_3)}{r_2} \right)
\end{aligned}$$

Collecting terms and replacing the compressible dynamic head, $0.5\rho V_{2j}^2$, the mixing loss coefficient for a radial, variable area mixing is given by eqn.(C.13):

$$\boxed{
\begin{aligned}
\bar{\omega}_{mix} = \frac{\Delta P_0}{P_{02} - P_2} = & \frac{P_2 \left(1 - \frac{r_2}{r_3}\right)}{P_{02} - P_2} + \frac{r_2}{r_3} \cos^2 \kappa_2 \left\{ \overline{b_{sep}}^2 \left[\frac{r_2}{r_3} \left(2 - \frac{1}{\cos^2 \beta_3} \right) - 8 \tan^2 \kappa_2 \frac{\left(1 - \frac{r_3}{r_2}\right)}{\left(1 + \frac{r_3}{r_2}\right)^2} \right] \right. \\
& \left. - 2 \overline{b_{sep}} + \frac{r_3}{r_2} \frac{1}{\cos^2 \kappa_2} \right\}
\end{aligned}
}$$

(C.13)

Appendix D

Derivation of governing equations with blockage in the relative FoR

Continuity

The continuity in Lagrangian form, using the material derivative D/Dt is written as [109]:

$$\frac{D\rho}{Dt} + \rho \nabla \cdot \mathbf{w} = 0$$

,where

$$\frac{D\rho}{Dt} = \frac{\partial \rho}{\partial t} + w_x \frac{\partial \rho}{\partial x} + w_\theta \frac{1}{r} \frac{\partial \rho}{\partial \theta} + w_r \frac{1}{r} \frac{\partial r \rho}{\partial r}$$

Multiplying by blockage, b :

$$b \frac{D\rho}{Dt} + b \rho \nabla \cdot \mathbf{w} = 0$$

Using the following identity:

$$\nabla \cdot ((b\rho) \mathbf{w}) = (b\rho) \nabla \cdot \mathbf{w} + (\mathbf{w} \cdot \nabla) (b\rho) \quad (\text{D.1})$$

or

$$(\mathbf{w} \cdot \nabla) (b\rho) = \nabla \cdot ((b\rho) \mathbf{w}) - (b\rho) \nabla \cdot \mathbf{w}$$

Noting that,

$$\frac{D\rho}{Dt} = \frac{\partial \rho}{\partial t} + (\mathbf{w} \cdot \nabla) \rho$$

The continuity in Lagrangian form multiplied by blockage gives:

$$b \frac{D\rho}{Dt} = \frac{b\partial\rho}{\partial t} + (\mathbf{w} \cdot \nabla)(\rho b) = \frac{b\partial\rho}{\partial t} + \nabla \cdot ((b\rho) \mathbf{w}) - (b\rho) \nabla \cdot \mathbf{w}$$

Since blockage is not a time-dependent quantity, it can be introduced inside the time derivative. The continuity equation with blockage in the relative FoR is given by eqn.(D.2):

$$\boxed{\frac{\partial b\rho}{\partial t} + \nabla \cdot (b\rho \mathbf{w}) = 0} \quad (\text{D.2})$$

Momentum

$$\rho \frac{D\mathbf{w}}{Dt} = -\nabla P + \rho \mathbf{f} - 2(\Omega \times \mathbf{w}) + \Omega \times (\Omega \times \mathbf{r})$$

Multiplying by the blockage:

$$b\rho \frac{D\mathbf{w}}{Dt} = -b\nabla P + b\rho \mathbf{f}$$

To implement the identity used before for the derivation of the continuity equation (eqn.(D.1)), the momentum equations are decomposed in the axial, circumferential and radial components.

Axial momentum:

$$\begin{aligned} \rho \frac{Dw_x}{Dt} &= -\frac{\partial P}{\partial x} + \rho f_x \\ \rho \frac{\partial w_x}{\partial t} + \rho \mathbf{w} \cdot \nabla w_x &= -\frac{\partial P}{\partial x} + \rho f_x \end{aligned}$$

Multiply with blockage:

$$b\rho \frac{\partial w_x}{\partial t} + b\rho \mathbf{w} \cdot \nabla w_x = -b \frac{\partial P}{\partial x} + b\rho f_x$$

Using the following identity to convert to conservation form:

$$\nabla \cdot (w_x (\rho \mathbf{w})) = (\rho \mathbf{w}) \cdot \nabla w_x + w_x \nabla \cdot (\rho \mathbf{w})$$

$$(\rho \mathbf{w}) \cdot \nabla w_x = \nabla \cdot (w_x (\rho \mathbf{w})) - w_x \nabla \cdot (\rho \mathbf{w})$$

Introducing blockage:

$$\nabla \cdot (w_x (b\rho \mathbf{w})) = (b\rho \mathbf{w}) \cdot \nabla w_x + w_x \nabla \cdot (b\rho \mathbf{w})$$

$$(b\rho \mathbf{w}) \cdot \nabla w_x = \nabla \cdot (w_x (b\rho \mathbf{w})) - w_x \nabla \cdot (b\rho \mathbf{w})$$

For the time derivative a similar approach is followed:

$$\begin{aligned} \frac{\partial (\rho w_x)}{\partial t} &= w_x \frac{\partial (\rho)}{\partial t} + \rho \frac{\partial (w_x)}{\partial t} \\ \rho \frac{\partial (w_x)}{\partial t} &= \frac{\partial (\rho w_x)}{\partial t} - w_x \frac{\partial (\rho)}{\partial t} \end{aligned}$$

Introducing blockage:

$$b\rho \frac{\partial w_x}{\partial t} = \frac{\partial (b\rho w_x)}{\partial t} - w_x \frac{\partial (b\rho)}{\partial t}$$

Substituting the above terms in the non-conservation form gives:

$$\begin{aligned} b\rho \frac{\partial w_x}{\partial t} + b\rho \mathbf{w} \cdot \nabla w_x &= -b \frac{\partial P}{\partial x} + b\rho f_x \\ \frac{\partial (b\rho w_x)}{\partial t} - w_x \frac{\partial (b\rho)}{\partial t} + \nabla \cdot (w_x (b\rho \mathbf{w})) - w_x \nabla \cdot (b\rho \mathbf{w}) &= -b \frac{\partial P}{\partial x} + b\rho f_x \\ \frac{\partial (b\rho w_x)}{\partial t} - w_x \frac{\partial (b\rho)}{\partial t} + \nabla \cdot (w_x b\rho \mathbf{w}) - w_x \nabla \cdot (b\rho \mathbf{w}) &= -b \frac{\partial P}{\partial x} + b\rho f_x \end{aligned}$$

Collecting terms and noting that the term inside the parenthesis is the continuity:

$$\frac{\partial (b\rho w_x)}{\partial t} + \nabla \cdot (w_x (b\rho \mathbf{w})) - w_x \left(\overbrace{\frac{\partial (b\rho)}{\partial t} + \nabla \cdot (b\rho \mathbf{w})}^{\text{continuity}=0} \right) = -b \frac{\partial P}{\partial x} + b\rho f_x$$

Which reduces to:

$$\frac{\partial (b\rho w_x)}{\partial t} + \nabla \cdot (w_x (b\rho \mathbf{w})) = -b \frac{\partial P}{\partial x} + b\rho f_x$$

To insert the blockage parameter into the pressure gradient term, the following substitution is performed:

$$\frac{\partial bP}{\partial x} = P \frac{\partial b}{\partial x} + b \frac{\partial P}{\partial x}$$

The axial momentum equation with blockage in the relative FoR is given by eqn.(D.3):

$$\boxed{\frac{\partial (b\rho w_x)}{\partial t} + \nabla \cdot (w_x (b\rho \mathbf{w})) = -\frac{\partial bP}{\partial x} + P \frac{\partial b}{\partial x} + b\rho f_x} \quad (\text{D.3})$$

Circumferential momentum:

$$\begin{aligned} \rho \frac{Dw_\theta}{Dt} &= -\frac{\partial P}{\partial \theta} - 2\rho\Omega w_r + \rho f_\theta \\ \rho \frac{\partial w_\theta}{\partial t} + \rho \mathbf{w} \cdot \nabla w_\theta &= -\frac{\partial P}{\partial \theta} - 2\rho\Omega w_r + \rho f_\theta \end{aligned}$$

Multiply with blockage:

$$b\rho \frac{\partial w_\theta}{\partial t} + b\rho \mathbf{w} \cdot \nabla w_\theta = -b \frac{\partial P}{\partial \theta} - 2b\rho\Omega w_r + b\rho f_\theta$$

Following the same procedure:

$$\begin{aligned} \rho \frac{\partial (w_\theta)}{\partial t} &= \frac{\partial (\rho w_\theta)}{\partial t} - w_\theta \frac{\partial (\rho)}{\partial t} \\ (b\rho \mathbf{w}) \cdot \nabla w_\theta &= \nabla \cdot (w_\theta (b\rho \mathbf{w})) - w_\theta \nabla \cdot (b\rho \mathbf{w}) \end{aligned}$$

Which reduces to:

$$\frac{\partial (b\rho w_\theta)}{\partial t} + \nabla \cdot (w_\theta b\rho \mathbf{w}) = -b \frac{\partial P}{\partial \theta} - 2b\rho\Omega w_r + b\rho f_\theta$$

To insert the blockage parameter into the pressure gradient term, the the following substitution is performed:

$$\frac{\partial bP}{\partial \theta} = P \frac{\partial b}{\partial \theta} + b \frac{\partial P}{\partial \theta}$$

The circumferential momentum equation with blockage in the relative FoR is given by eqn.(D.4):

$$\boxed{\frac{\partial (b\rho w_\theta)}{\partial t} + \nabla \cdot (b\rho w_\theta \mathbf{w}) = -\frac{\partial bP}{\partial \theta} + P \frac{\partial b}{\partial \theta} - 2b\rho\Omega w_r + b\rho f_\theta} \quad (\text{D.4})$$

In turbomachinery applications, blockage is uniform in the circumferential direction, thus $P \frac{\partial b}{\partial \theta} = 0$.

Radial momentum

$$\begin{aligned}\rho \frac{Dw_r}{Dt} &= -\frac{\partial P}{\partial r} + 2\Omega w_\theta \rho + \rho \Omega^2 r + \rho f_r \\ \rho \frac{\partial w_r}{\partial t} + \rho \mathbf{w} \cdot \nabla w_r &= -\frac{1}{r} \frac{\partial P}{\partial r} + 2\Omega w_\theta \rho + \rho \Omega^2 r + \rho f_r\end{aligned}$$

Multiply with blockage:

$$b\rho \frac{\partial w_r}{\partial t} + b\rho \mathbf{w} \cdot \nabla w_r = -b \frac{1}{r} \frac{\partial P}{\partial r} + 2b\rho \Omega w_\theta + b\rho \Omega^2 r + b\rho f_r$$

Following the same procedure:

$$\begin{aligned}\rho \frac{\partial (w_r)}{\partial t} &= \frac{\partial (\rho w_r)}{\partial t} - w_r \frac{\partial (\rho)}{\partial t} \\ (b\rho \mathbf{w}) \cdot \nabla w_r &= \nabla \cdot (w_r (b\rho \mathbf{w})) - w_r \nabla \cdot (b\rho \mathbf{w})\end{aligned}$$

Which reduces to:

$$\frac{\partial (b\rho w_r)}{\partial t} + \nabla \cdot (b\rho w_r \mathbf{w}) = -b \frac{1}{r} \frac{\partial P}{\partial r} + 2b\rho \Omega w_\theta + b\rho \Omega^2 r + b\rho f_r$$

To insert the blockage parameter into the pressure gradient term, the following substitution is performed:

$$\frac{\partial brP}{\partial r} = Pr \frac{\partial b}{\partial r} + br \frac{\partial P}{\partial r} + bP \frac{\partial r}{\partial r} = Pr \frac{\partial b}{\partial r} + br \frac{\partial P}{\partial r} + bP$$

The radial momentum equation with blockage in the relative FoR is given by eqn.(D.5):

$$\boxed{\frac{\partial (b\rho w_r)}{\partial t} + \nabla \cdot (w_r b\rho \mathbf{w}) = -\frac{\partial brP}{\partial r} + P \frac{\partial b}{\partial r} + b \frac{P}{r} + 2b\rho \Omega w_\theta + b\rho \Omega^2 r + b\rho f_r} \quad (\text{D.5})$$

Energy:

$$\rho \frac{D(I)}{Dt} - \frac{\partial P}{\partial t} = 0$$

Or

$$\rho \frac{\partial(I)}{\partial t} + \rho (\mathbf{w} \cdot \nabla) I - \frac{\partial P}{\partial t} = 0$$

Multiply with blockage:

$$b\rho \frac{\partial(I)}{\partial t} + b(\mathbf{w} \cdot \nabla) I - b \frac{\partial P}{\partial t} = 0$$

Following the same procedure:

$$b\rho \frac{\partial(I)}{\partial t} = b\rho \frac{\partial(I)}{\partial t} - I \frac{\partial(b\rho)}{\partial t}$$

$$(b\rho \mathbf{w}) \cdot \nabla I = \nabla \cdot (I (b\rho \mathbf{w})) - I \nabla \cdot (b\rho \mathbf{w})$$

Which finally gives:

$$\frac{\partial(b\rho I)}{\partial t} + \nabla \cdot (b\rho \mathbf{w} I) - b \frac{\partial P}{\partial t} = 0$$

To insert the blockage parameter into the pressure gradient term, the the following substitution is performed:

$$\frac{\partial bP}{\partial t} = P \frac{\partial b}{\partial t} + b \frac{\partial P}{\partial t}$$

$$\frac{\partial(b\rho I)}{\partial t} + \nabla \cdot (b\rho \mathbf{w} I) - \frac{\partial bP}{\partial t} + P \overbrace{\frac{\partial b}{\partial t}}^0 = 0$$

The energy equation with blockage in the relative FoR is given by eqn.(D.6):

$$\boxed{\frac{\partial(b\rho E_r)}{\partial t} + \nabla \cdot (b\rho \mathbf{w} I) = 0} \quad (\text{D.6})$$

Appendix E

Derivation of Cubic Shape Function

The cubic shape functions using the normalised meridional coordinate, ξ can be expressed by a third order polynomial as follows:

$$u(\xi) = c_1 + c_2\xi + c_3\xi^2 + c_4\xi^3$$

The normalised meridional coordinate, ξ , is given by:

$$\xi = \frac{m - m_{LE}}{m_{TE} - m_{LE}}$$

The control points of the shape function are: $\xi = [0, \xi_2, \xi_3, 1]$. The boundary conditions at the control points are:

For $\xi = 0$:

$$c_1 = u_1 \tag{E.1}$$

For $\xi = \xi_2$:

$$u_2 = c_1 + c_2\xi_2 + c_3\xi_2^2 + c_4\xi_2^3$$

Substituting eqn.(E.1) in the above equation gives:

$$u_2 = u_1 + c_2\xi_2 + c_3\xi_2^2 + c_4\xi_2^3 \tag{E.2}$$

For $\xi = 1$:

$$u_4 = c_1 + c_2 + c_3 + c_4$$

Substituting eqn.(E.1) in the above equation gives:

$$u_4 = u_1 + c_2 + c_3 + c_4 \tag{E.3}$$

For the 4th constraint, instead of forcing the polynomial to pass through the third control point, ξ_3 , a derivative constraint is imposed at $\xi = \xi_2$ as follows:

$$u_2' = \left. \frac{\partial u}{\partial \xi} \right|_{\xi_2} = c_2 + 2c_3\xi_2 + 3c_4\xi_2^2 \quad (\text{E.4})$$

From eqn.(E.4):

$$c_2 = u_2' - 2c_3\xi_2 - 3c_4\xi_2^2$$

Combining eqns.(E.3) and (E.4):

$$u_4 - u_1 - c_3 - c_4 = u_2' - 2c_3\xi_2 - 3c_4\xi_2^2$$

$$c_3 = \frac{u_2' - 3c_4\xi_2^2 - u_4 + u_1 + c_4}{(2\xi_2 - 1)}$$

The coefficient, c_3 is given by eqn.(E.6) as a function of c_4 :

$$c_3 = \frac{u_2' - u_4 + u_1 - c_4(3\xi_2^2 - 1)}{(2\xi_2 - 1)} \quad (\text{E.5})$$

From eqn.(E.2):

$$\frac{u_2}{\xi_2} - \frac{u_1}{\xi_2} - c_3\xi_2 - c_4\xi_2^2 = c_2$$

From eqn.(E.3):

$$u_4 - u_1 - c_3 - c_4 = c_2$$

Combining the above equations gives:

$$\frac{u_2}{\xi_2} - \frac{u_1}{\xi_2} - c_3\xi_2 - c_4\xi_2^2 = u_4 - u_1 - c_3 - c_4$$

$$\frac{u_2}{\xi_2} - \frac{u_1}{\xi_2} - u_4 + u_1 = -c_3 + c_3\xi_2 - c_4 + c_4\xi_2^2$$

$$\frac{u_2 - u_1}{\xi_2} - u_4 + u_1 = c_3(\xi_2 - 1) + c_4(\xi_2^2 - 1)$$

The coefficient c_3 is computed as a function of c_4 coefficient as per eqn.(E.6).

$$c_3 = \frac{\frac{u_2 - u_1}{\xi_2} - u_4 + u_1}{(\xi_2 - 1)} - c_4(\xi_2 + 1) \quad (\text{E.6})$$

Equating the 2 equations for c_3 gives:

$$\frac{\frac{u_2 - u_1}{\xi_2} - u_4 + u_1}{(\xi_2 - 1)} - c_4(\xi_2 + 1) = \frac{u_2' - u_4 + u_1 - c_4(3\xi_2^2 - 1)}{(2\xi_2 - 1)}$$

Collecting terms:

$$(2\xi_2 - 1) \frac{\frac{u_2 - u_1}{\xi_2} - u_4 + u_1}{(\xi_2 - 1)} - u_2' + u_4 - u_1 = c_4 \left[(\xi_2 + 1)(2\xi_2 - 1) - (3\xi_2^2 - 1) \right]$$

The coefficient, c_4 is finally given by eqn.(E.7):

$$c_4 = \frac{(2\xi_2 - 1) \frac{\frac{u_2 - u_1}{\xi_2} - u_4 + u_1}{(\xi_2 - 1)} - u_2' + u_4 - u_1}{(\xi_2 + 1)(2\xi_2 - 1) - (3\xi_2^2 - 1)} \quad (\text{E.7})$$

From eqn.(E.3) the coefficient c_2 can now be determined:

$$c_2 = u_4 - u_1 - c_3 - c_4 \quad (\text{E.8})$$

And the coefficient c_1 is computed as per eqn.(E.1):

$$c_1 = u_1 \quad (\text{E.9})$$