

CRANFIELD UNIVERSITY

MAURO RIGHI

THREE-DIMENSIONAL THROUGH-FLOW MODELLING OF
AXIAL COMPRESSOR ROTATING STALL AND SURGE

SCHOOL OF AEROSPACE, TRANSPORT AND
MANUFACTURING
CENTRE FOR PROPULSION ENGINEERING

FULL TIME PhD

Academic Year: 2014-2017

Supervisor: Prof. V. Pachidis, Dr. L. Könözy
January 2018

CRANFIELD UNIVERSITY

SCHOOL OF AEROSPACE, TRANSPORT AND
MANUFACTURING
CENTRE FOR PROPULSION ENGINEERING

FULL TIME PhD

Academic Year 2014-2017

MAURO RIGHI

Three-Dimensional Through-Flow Modelling of Axial Compressor
Rotating Stall And Surge

Supervisor: Prof. V. Pachidis, Dr. L. Könözy
January 2018

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

© Cranfield University 2017. All rights reserved. No part of this
publication may be reproduced without the written permission of the
copyright owner.

Executive summary

The operation of an aero-engine is limited by the occurrence of compressor stall, and compressor performance is sacrificed to maintain a sufficient margin of operation. Compressor stall also plays an important part in the event of a shaft failure, and in determining if this will result in a rotor burst. In Cranfield University a tool to model the whole engine during shaft failure has been developed, but it requires the knowledge of the compressor performance during stall.

Low order 1D, 2D or 3D methods to model compressor stall exist in literature, but they are still at a low maturity level and not applicable for commercial use. The only methods available are expensive experimental testing and transient 3D CFD, which has unacceptable computational costs. The objective of this PhD project has therefore been identified in the development of a fast and robust 3D tool to model compressor stall, and in its validation with data from low-speed experimental rigs.

The tool created is a three-dimensional through-flow code which uses empirical correlations to model the blade row performance. A novel methodology has been developed to estimate the performance of blade rows in reverse flow, based on previous models of separated blade passages. The Godunov scheme has been chosen to create the 3D, unsteady, cylindrical, compressible, finite volume method Euler solver on which the code is based. Appropriate body forces and boundary conditions have been chosen and implemented.

The validation carried out on two low-speed compressors demonstrates the applicability of the proposed formulation, with successful prediction of the performance during reverse flow, rotating stall and surge. The speed, size and structure of the rotating stall cell have been successfully matched to experimental data. The developed tool can reproduce the forward flow, reverse and rotating stall regions of the map in less than 72 hours, at a computational speed unrivalled by modern commercial CFD codes.

Acknowledgements

I would like to thank first of all my supervisor Prof. Pachidis, for the possibility of undertaking this research in the Cranfield UTC and having entrusted me with this challenging, yet deeply rewarding, topic. Thank for your guidance, your support, your patience and having believed in me even when I didn't. I would also like to thank Dr. Konozy for having joined this research with such enthusiasm; thanks for all your technical guidance and the long chats.

I would like to thank Rolls-Royce plc for having sponsored this research. In particular I would like to thank my industrial advisors Arthur Rowe, for sharing even a bit of his vast knowledge with me, and Richard Tunstall, for all his guidance throughout the project. Many thanks also to Chris Sheaf and Stephen Brown for their invaluable contributions.

I would also like to thank Prof. MacManus, and Dr. Zachos for their technical advices. One more thanks goes to Prof. Day of Cambridge, for having dug out for me the geometries of one of my two main test cases and pointing out where to find the details of the other.

Thanks to all my friends and colleagues in the department. In particular to Giovanni, Alvisè and Enrico for all the time spent together, all our adventures and all our discussions, technical and not. Thanks to my research team, to Lucas and Ibrahim for their technical help and support. Thanks to Chiara and Riccardo for having been more than housemates, but true friends, and having looked after me and cooked for me. Thanks to Rafał, for all his patience, support and love.

Last but not least I would like to thank my family, my parents and my brother who, even from far away, have always supported me during this project.

Contents

<i>Executive summary</i>	i
<i>Acknowledgements</i>	iii
<i>List of figures</i>	ix
<i>List of Tables</i>	xvii
<i>List of symbols and acronyms</i>	xix
1. <i>Introduction</i>	1
1.1 Compressor stall	1
1.2 Shaft failure and turbine over-speed	7
1.3 Project aim and objectives	10
1.4 Thesis outline	11
2. <i>Project overview</i>	13
2.1 Contribution to knowledge	14
2.2 Industrial relevance	15
3. <i>Literature review</i>	17
3.1 Compressor stall theory	17
3.1.1 Stall inception	17
3.1.2 Compressor stall types	20
3.1.3 Stalled performance characterization	26
3.1.4 Compressor stall dynamics and stability	29
3.1.5 Compressor stall recoverability	36
3.2 Analytical/numerical modelling of compressor stall	37
3.2.1 0D/1D models	38

3.2.2	2D models	42
3.2.3	3D models	46
3.2.4	3D CFD	50
3.2.5	Compressor stall modelling methods summary	52
3.2.6	Full engine stall modelling	54
3.3	Experimental investigation of compressor stall	58
3.3.1	Eastland and Gamache's test on the 3-stage compressor	58
3.3.2	Day's test on the 4-stage compressor	61
3.4	Summary	62
4.	<i>Methodology</i>	65
4.1	Tool development overview	65
4.2	Euler solver	66
4.2.1	Monotone and TVD scheme	66
4.2.2	The Godunov scheme	68
4.2.3	Grid generation	71
4.2.4	Equations	71
4.2.5	Solver implementation	75
4.3	Body force method	77
4.3.1	Pseudo-meridional planes	77
4.3.2	Regimes	79
4.3.3	Direction imposed	79
4.3.4	Body force formulation	81
4.4	Empirical correlations	84
4.5	Forward flow correlations	86
4.5.1	Pressure loss coefficients	87
4.5.2	Deviation	90
4.5.3	Rolls-Royce correlations	91
4.6	Stall detection	92
4.7	Separated flow modelling	93
4.8	Boundary conditions	96

4.8.1	Non-reflective boundary conditions	96
4.8.2	Wall boundary conditions	100
4.8.3	Plenum and throttle model	101
4.9	Simulation input, output and strategy	103
4.9.1	Simulation input	104
4.9.2	Simulation output	107
4.9.3	Simulation strategy	107
4.10	Programming	110
4.11	Summary	112
5.	<i>Test case geometries and set-up</i>	115
5.1	2-stage NASA rig	115
5.2	3-stage MIT rig	116
5.3	4-stage C106 rig	118
5.4	Modern high-speed HP compressor	118
5.5	Non-dimensionalization of characteristics and transients	119
6.	<i>Results and discussion</i>	123
6.1	Grid dependency study	123
6.2	Three-dimensional Euler solver validation	128
6.3	Forward flow validation	133
6.4	Reverse flow validation	137
6.4.1	3-stage MIT rig	137
6.4.2	Modern high-speed HP compressor	141
6.5	Rotating stall validation	147
6.5.1	3-stage MIT rig	148
6.5.2	4-stage C106 rig	168
6.6	Surge simulation	173
6.6.1	3-stage MIT rig	173
6.6.2	4-stage C106 rig	179
6.6.3	2D and 3D surge simulations	184

7. <i>Conclusions and future development</i>	191
7.1 Summary	191
7.2 Contribution to knowledge	193
7.3 Recommendations for future work	194
<i>References</i>	197
<i>Appendices</i>	205
A Formulation of the solver	207
B Formulation of the reverse flow deviation correlation	210
C Formulation of the non-reflective boundary condition	212
D Code flow chart	215

List of figures

1.1	Sukhoi Su-57 prototype suffering a compressor stall at MAKS 2011.	1
1.2	Example of a typical compressor map.	3
2.1	Project roadmap.	13
3.1	Stage total-to-static pressure rise characteristic of a multi-stage compressor at different operating conditions.	18
3.2	Waves from a modal stall inception visible in measurements of axial flow coefficient at various circumferential positions.	19
3.3	Different operating point at which modal and spike inception occur.	20
3.4	Stall cell developing into a mature rotating stall.	21
3.5	Passive and active stall cell models.	22
3.6	Effect of the compressor geometry on the stall cell structure.	23
3.7	Stall cell propagation mechanism.	23
3.8	Surge cycle trajectories for different values of downstream volume.	25
3.9	Example of complete compressor map.	26
3.10	Stall trajectory simulated with a 2D lumped model, with different downstream volumes.	28
3.11	Complete compressor map with throttle characteristic.	28
3.12	Surge trajectory for different volumes.	30
3.13	Trajectories of classic surge and mature rotating stall development.	30
3.14	Stall inception behaviour of the Rolls-Royce VIPER compressor at different speeds.	31
3.15	Mass-spring-damper system analogy of the Moore and Greitzer lumped model.	33
3.16	Equilibrium points of a compressor rig at different throttle settings	34
3.17	Transient responses of surge system to a pressure perturbation.	35

3.18	Hysteresis in rotating stall and effect on recoverability.	36
3.19	Compressor stall models in order of complexity and computational cost.	37
3.20	Sources of pressure loss and deviation in a cascade row.	38
3.21	Zero-dimensional lumped model created by Greitzer.	39
3.22	C106 complete compressor map with results from FENICE.	43
3.23	Simulated post-stall behaviours of the Rolls-Royce VIPER compressor.	45
3.24	Example of compressor and plenum domain as modelled by Escuret and Garnier.	48
3.25	Flow separated at a lengthscale smaller than the grid element.	49
3.26	Surge and rotating stall of Gamache's 3-stage compressor modelled by Longley.	51
3.27	Part-span stall cells development, following a stator vane restaggering.	52
3.28	Compressor/combustor domain of the shaft failure modelling tool.	55
3.29	Components of the code WEST.	56
3.30	The 3-stage low-speed compressor tested at MIT.	59
3.31	A schematic of Gamache's test rig.	60
3.32	Hot wire trace during rotating stall.	61
3.33	The 4-stage compressor tested by Day.	61
3.34	Absolute velocity measured at compressor outlet during surge	62
4.1	Monotonic behaviour of a numerical scheme.	67
4.2	Riemann problem at an inter-volume boundary.	68
4.3	Interaction of waves from different boundaries.	70
4.4	Volumes' vertexes and centroids with the indexing used.	71
4.5	Example of grid generation for the compressor rig described in NASA TP1493.	72
4.6	Areas nomenclature for a generic grid element.	75
4.7	Flux calculation across an horizontal area.	75
4.8	Real and virtual blade passages in the tool.	78
4.9	Meridional view of the pseudo-plane in a rotor blade row.	78

4.10	Axial and radial flow projection on the local pseudo-plane.	78
4.11	Possible flow regimes.	79
4.12	Components of the imposed direction β	80
4.13	]	81
4.14	Direction of turning and pressure loss forces in the relative FOR. . . .	82
4.15	Separated cell shed from the leading edge during passage stalling. . .	85
4.16	Example of stalled virtual passages in a rotor blade row and their translation.	86
4.17	Nomenclature of angles and dimensions in a blade passage.	87
4.18	Loss coefficient profile and the effect of Mach number.	89
4.19	Interpolation of pressure loss coefficient.	90
4.20	Separated flow structure within a typical blade passage in reverse flow.	94
4.21	]	95
4.22	Boundary conditions applied to the compressor's computational do- main.	97
4.23	Types of boundary conditions and their dynamic response to a pres- sure wave exiting the domain.	98
4.24	Ghost nodes at the hub wall.	100
4.25	Trajectory of rotating stall operating point with throttle opening and closure.	110
4.26	Strategy used to test the post-stall behaviour of a new compressor design.	110
4.27	Computational time reduction with number of threads employed. . .	112
5.1	Meridional view of the NASA 2-stage rig.	116
5.2	Blades from the 3-stage MIT rig.	117
5.3	Example of operating point calculated at domain inlet or outlet. . . .	121
6.1	Values of axial flow coefficient and pressure rise coefficient with the increasing grid refinement	126
6.2	Three-dimensional flow modelled using ANSYS FLUENT.	129
6.3	Annulus used in the Euler solver validation.	129

6.4	Extrapolated errors from the grid convergence study for the Euler solver test case	131
6.5	Comparison of axial and circumferential velocity modelled with the new tool and ANSYS FLUENT TM	132
6.6	Comparison of static pressure modelled along the three lines and at constant radius at inlet.	133
6.7	Extrapolated errors from the grid convergence study for the 2-stage NASA rig case	134
6.8	Comparison of experimental and modelled compressor map of the NASA 2-stage rig.	135
6.9	Comparison of experimental and modelled temperature ratio map of the NASA 2-stage rig.	136
6.10	Reverse characteristic validation on the 3-stage MIT rig.	137
6.11	Reverse flow torque characteristic validation for the 3-stage MIT rig.	138
6.12	Span distribution of fluid in the meridional plane during reverse flow.	139
6.13	Modelled flow-field in the meridional plane with over-imposed axial flow coefficient; $\phi = -0.1$	140
6.14	Modelled flow-field in the meridional plane with over-imposed axial flow coefficient; $\phi = -0.05$	140
6.15	Variation of extrapolated relative error with grid refinement in the modern high-speed HP compressor case.	143
6.16	Reverse flow characteristics validation on the 6-stage high-speed HPC compressor, using a simplified body-force method.	145
6.17	Reverse flow temperature characteristics validation on the 6-stage high-speed HPC compressor, using a simplified body-force method.	145
6.18	Reverse flow characteristics validation on the 6-stage high-speed HPC compressor, using the body force method with correction for compressibility.	146
6.19	Reverse flow temperature characteristics validation on the 6-stage high-speed HPC compressor, using the body force method with correction for compressibility	146

6.20	Different patterns used at distortion inlet.	148
6.21	Trajectories on the map of two rotating stall developments with dif- ferent throttle closure schedules.	151
6.22	3-stage MIT rig complete map validation.	152
6.23	Operating point trajectory during two consecutive throttle openings. .	152
6.24	Rotating stall torque characteristic validation on the 3-stage MIT rig full map.	154
6.25	Flow coefficient traces at different circumferential positions.	155
6.26	Comparison of modelled and experimental rotating stall speed of the 3-stage MIT rig.	156
6.27	Comparison of modelled and experimental stall cell size with examples of cell shape.	156
6.28	Axial Mach number at half span (radius $r=0.285$ m) during rotating stall.	158
6.29	Static to static pressure rise coefficient at half span (radius $r=0.285$ m) during rotating stall.	159
6.30	Flow direction projected at half span (radius $r=0.285$ m) during ro- tating stall.	160
6.31	Description of the flow direction across the stages in a stall cell as reported by Day and Cumpsty.	160
6.32	Temperature rise coefficient at half span (radius $r=0.285$ m) during rotating stall.	162
6.33	Flow angle in the meridional plane at half span (radius $r=0.285$ m) during rotating stall.	163
6.34	Axial Mach number at IGV inlet during rotating stall.	164
6.35	Axial Mach number contours on the annulus section at different blade rows' inlet. Case at $\phi = 0.11$	165
6.36	Axial Mach number contours on the annulus section at different axial positions. Case at $\phi = 0.38$	166
6.37	Axial Mach number contours on the annulus section at different axial positions. Case at $\phi = 0.475$	167

6.38	Simulated rotating stall shape close to zero mass-flow.	168
6.39	Extrapolated errors from the grid convergence study for the 4-stage C106 rig test case	170
6.40	Full compressor map validation on the 4-stage C106 rig.	171
6.41	Rotating stall cell speed modelled on the 4-stage C106 rig.	172
6.42	Rotating stall cell size modelled on the 4-stage C106 rig.	173
6.43	Stall trajectories at varying plenum volume on the 3-stage MIT rig. .	175
6.44	Stall trajectories with volume of 3 m^3 , with axial Mach number at 2nd rotor inlet display the flow structure at different instants during surge.	176
6.45	Stall trajectories with different inlet distortions on the 3-stage MIT rig.	178
6.46	Axial Mach number at IGV inlet during surge transient, with different inlet distortions.	178
6.47	Comparison of surge profiles obtained with different throttle closure, $V_{Pl} = 3 \text{ m}^3$	180
6.48	Comparison of experimental and modelled surge frequency for differ- ent plenum volumes.	181
6.49	Comparison of experimental and modelled pressure profiles during a surge cycle.	182
6.50	ψ profiles measured at plenum and compressor exit during surge. . . .	183
6.51	Surge cycle of the 3-stage MIT rig, modelled with symmetry assump- tion and full 3D model. $V_{Pl} = 3 \text{ m}^3$	185
6.52	Pressure rise profile during surge of the 3-stage MIT rig, modelled with symmetry assumption and full 3D model. $V_{Pl} = 3 \text{ m}^3$	186
6.53	Surge cycle of the 3-stage MIT rig, modelled with symmetry assump- tion and full 3D model. $V_{Pl} = 1.5 \text{ m}^3$	186
6.54	Pressure rise profile during surge of the 3-stage MIT rig, modelled with symmetry assumption and full 3D model. $V_{Pl} = 1.5 \text{ m}^3$	187
6.55	Operating point trajectory during stall triggered with throttle closure, modelled with symmetry assumption and full 3D model. Test case: 3-stage MIT rig, $V_{Pl} = 1.0 \text{ m}^3$	188

6.56	Meridional flow structure in a failed simulation in which rotating stall was not modelled because of the symmetry assumption.	188
6.57	Operating point trajectory during stall triggered with throttle closure, modelled with symmetry assumption. Test case: 3-stage MIT rig, $V_{Pl} = 0.85 \text{ m}^3$	189
D.1	Code flow chart.	215

List of Tables

1.1	Typical stall margin budget, from Aerospace Recommended practice ARP 1420.	6
3.1	Comparison of rotating stall and surge models in literature.	53
6.1	Number of elements, element size, time step and computational cost of the cases used in the grid dependency study.	124
6.2	Results of the grid dependency study.	127
6.3	Simulation parameters for the Euler solver validation.	130
6.4	Grid convergence study for the 3D Euler solver case	131
6.5	Comparison of integral results from the new tool and ANSYS FLUENT TM .132	
6.6	Grid convergence study for the the 2-stage NASA rig case	134
6.7	Grid convergence study for the modern high-speed HP compressor, case at low speed	142
6.8	Grid convergence study for the modern high-speed HP compressor, case at high speed	143
6.9	Parameters of the two rotating stall development simulations with different throttle closing schedules.	150
6.10	Number of elements, element size, time step and computational cost of the grids used in the grid dependency study for the 4-stage C106 rig test case.	169
6.11	Grid convergence study for the 4-stage C106 rig test case	170
6.12	Parameters of the C106 rotating stall simulations.	171
6.13	Parameters of the 3-stage MIT rig surge simulations with variable plenum volume.	174
6.14	Parameters of the 3-stage MIT rig surge simulations with different inlet distortions	177

6.15 Parameters of the 4-stage C106 rig surge simulations.	181
--	-----

List of symbols and acronyms

a	Speed of sound [m/s]
A	Area [m ²]
A_{RSC}	Rotating stall cell area [m ²]
$\Delta A_{\%}$	Throttle area variation [-]
b	Longley's blockage parameter [-]
B	Greitzer's parameter [-]
c	Chord [m]
C_V, C_P	Specific heat at constant volume, pressure [J/(kg K)]
$C_{\omega TUNING}, C_{\delta TUNING}$	Empirical correlations tuning factor [-]
$d_1, d_2, \dots, d_5$	Non reflective BC coefficients
D, D_{EQ}	Diffusion factor, Equivalent diffusion factor [-]
DIS	Relative discrepancy [-]
$DC60$	60° sector distortion coefficient
e_{ex}	Extrapolated relative error [-]
e_a	Approximate relative error [-]
E	Radial fluxes
E_0	Specific internal energy [J/kg]
F	Volumetric body force [N/m ³]
f	Body force vector

F	Circumferential fluxes
G	Axial fluxes
<i>GCI</i>	Grid convergence index [-]
<i>h</i>	Annulus height [m], Grid element size [m]
<i>H₀</i>	Enthalpy [J/kg]
<i>i</i>	Incidence angle [rad], axial index [-]
<i>i_S</i>	Positive stall incidence angle [rad]
<i>i_{NS}</i>	Negative stall incidence angle [rad]
<i>I_{SW}</i>	Algorithm switch
<i>j</i>	Radial index [-]
<i>k</i>	Circumferential index [-]
<i>K</i>	Constant (various)
<i>L</i>	Length [m], Characteristic waves amplitudes
<i>m, m'</i>	Mass-flow, Corrected mass-flow [kg/s]
<i>M</i>	Mach number [-]
<i>n</i>	Apparent order of the numerical scheme [-]
<i>N</i>	Number of elements [-]
<i>P</i>	Pressure [Pa]
<i>P₀</i>	Total pressure [Pa]
<i>PR</i>	Pressure ratio [-]
ΔP_0	Local total pressure loss [Pa]
<i>Q</i>	Momentum [kg m/s]

r	Radius [m], Grid refinement factor [-]
R	Gas constant [$\frac{J}{kgK}$], Angle stall margin [rad]
R_{CUR}	Local radius of curvature [m]
s	Blade pitch [m]
S	Entropy [J/(kg K)]
S	Source terms
t	Time [s]
t_b	Blade thickness [m]
$\mathcal{T}$	Torque [N · m]
T	Temperature [K]
TR	Temperature ratio [-]
T_0	Total temperature [K]
t_{RSC}	Rotating stall cell passage period [s]
T_{RSC}	Rotating stall period [s]
u	Flow velocity in absolute frame of reference [m/s]
U	Blade speed [m/s]
U_{RSC}	Rotating stall cell translational speed [m/s]
U	Conservative variables
v_{RE}	Equivalent relative velocity ratio [m/s]
V	Volume [m ³]
V	Primitive variables
w	Flow velocity in relative frame of reference [m/s]

W_{EX}	Work exchanged [W/m ³]
x	Axial position [m]
X	Generic variable
α	Flow angle [rad]
β	Direction imposed by the body forces [rad]
γ	Specific heats ratio [-], Stagger angle [rad]
δ	Deviation angle [rad]
θ	Circumferential position [rad], camber angle [rad]
λ	Pseudo-meridional plane angle [rad], attack angle [rad]
ρ	Density [kg/m ³]
σ	Blade solidity [-]
τ	Time lag constant [s]
ψ	Pressure rise coefficient[-]
ϕ	Flow coefficient [-]
ψ_{τ}	Torque coefficient [-]
ω	Total pressure loss coefficient [-]
ψ_{TEMP}	Temperature rise coefficient [-]
Ω	Shaft rotational speed [rad/s]

SUBSCRIPTS

ATM	Atmospheric property
-------	----------------------

C	Compressor property
$CRIT$	Critical value
EW	Endwall losses
i	Element's radial index
IN	Blade passage's flow inlet
j	Element circumferential index, jet condition
k	Element axial index
N	Turning force
OUT	Blade passage's flow outlet
PL	Pressure loss force
Pl	Plenum
REF	Reference value
r	Radial direction
SS	Steady state
$S - S$	Static to static
T	Throttle
$T - T$	Total to total
x	Axial direction
η	Parallel to the imposed direction
θ	Circumferential direction
ξ	Pseudo-meridional direction
ψ	Normal to the imposed direction

$\star$	Design value
1	Upstream reverse flow
2	Fully mixed-out reverse flow

SUPERSCRIPTS & ACCENTS

$*$	Design value
$\bar{x}$	Average value of x
H	Value at horizontal face
θ	Value at circumferential face
V	Value at vertical face
t	Value at time t
RS	Values based on the Riemann problem solution

ACRONYMS

BC	Boundary conditions
CFL	Courant-Friedrich-Lewy's number
CFD	Computational fluid dynamics
DLL	Dynamic link library
EEC	Electronic engine control
FADEC	Full authority digital engine control
FOD	Foreign object damage

FOR	Frame of reference
FVM	Finite volume method
HLLC	Harten-Lax-van Leer-Contact
HP	High pressure
IGV	Inlet guide vane
IP	Intermediate pressure
LE	Leading edge
LHS	Left hand side
NGV	Nozzle guide vane
NRB	Non reflective boundary conditions
PLA	Power level angle
RANS	Reynold averaged Navier Stokes
RHS	Right hand side
RPM	Rotations per minute
SAS	Secondary air system
SLC	Streamline curvature
SST	Stage stacking technique
TE	Trailing edge
TVD	Total variation diminishing
VIGV	Variable inlet guide vane
VSV	Variable guide vane

Chapter 1

Introduction

1.1 Compressor stall

The operation of a jet engine is limited by the insurgence of a phenomenon called *compressor stall*. With this term we refer to the disruption of the airflow inside of the engine's compressor and the resulting loss of compressing capability. The effect of this stall can vary from a small momentary reduction in power, to a persistent loss in power, to a total loss of propulsion and serious damage to the engine.



Fig. 1.1: Sukhoi Su-57 prototype suffering a compressor stall at MAKS 2011 [1].

When compressor stall occurs this can often be observed by passengers and ground spectators as flames coming out of the engine's nozzle accompanied by a series of loud bangs, (figure 1.1). Although this is often referred to as 'explosions'

in the press, the flames are actually caused by the chaotic flow delivered to the combustion chamber by the stalled compressor. The irregular fuel distribution in the chamber causes some large droplets of kerosene to remain uncombusted all the way to the end of the turbine section and burn in the nozzle, causing the blaze. The loud bangs are the result of periodic pressure jumps at the engine intake and nozzle which cause sonic waves. Though no real explosion takes place, the occurrence of compressor stall causes serious damage to the engine and an expensive rebuild or repair is then necessary.

When an engine stalls during flight it often has to be switched off for the rest of the mission and this leads to large economic losses to the airlines due to the flight disruption (flying with only one engine, flying at reduced efficiency, emergency landing, flight delays, etc.). Furthermore the occurrence of compressor stall always leads to a loss of propulsion, if this occurs during critical parts of the mission (take-off, landing, go-around, etc.) it can put the safety of the passenger and crew at risk. As such, compressor stall is generally regarded as “something which must be avoided at all costs” [2].

Compressor stall occurs as a result of very specific engine operating conditions. A jet engine compressor is a turbo-machinery component which compresses the airflow coming from the intake and delivers the pressurised air to the combustion chamber. In stable conditions a pressure gradient is present across the compressor between the pressurised air in the back and the inlet flow; the compressor imposes an axial force on the airflow and balances the pressure gradient. This force is produced by the rotor and stator blade rows which impose a turning on the flow. In the same way that airfoils have a maximum lift they can produce before stalling, also the blade passages of a compressor have a maximum value of flow turning they can produce beyond which the flow separates. If the flow separates locally, the blades will fail to produce enough force to balance the pressure gradient and the compressor will become unstable. The locus of the operating conditions at which separation occurs forms a line in a compressor map called *surge line* which effectively is an upper limit to the area of stable operation, figure 1.2. The compressor becomes unstable whenever the operating point crosses the surge line. The difference in pressure

ratio between the point at which a constant speed-line crosses the surge line and the normal operating line is called *surge margin*. The ‘distance’ from the surge line could also be calculated as a mass-flow difference. However in an engine the pressure is measured directly by the probes while mass-flow can only be obtained from other measurements, for this reason the pressure ratio is the preferred method to quantify the distance from the stability limit. A parameter to quantify the surge margin is for example the term Z used in TURBOMATCH^{TM} , a modular code for calculation of gas turbine performances developed in Cranfield University [3]. The term Z is defined as

$$Z = \frac{PR^* - PR_{LOW}}{PR_{HIGH} - PR_{LOW}} \quad (1.1)$$

where PR^* is the design pressure ratio and PR_{HIGH} and PR_{LOW} are the maximum and minimum value of pressure ratio of that specific speed-line. While Z in eq. 1.1 refers to surge margin in design conditions, it can also be used during off-design calculations to quantify the proximity to stall; when Z reaches 1 the compressor becomes unstable and stall is triggered.

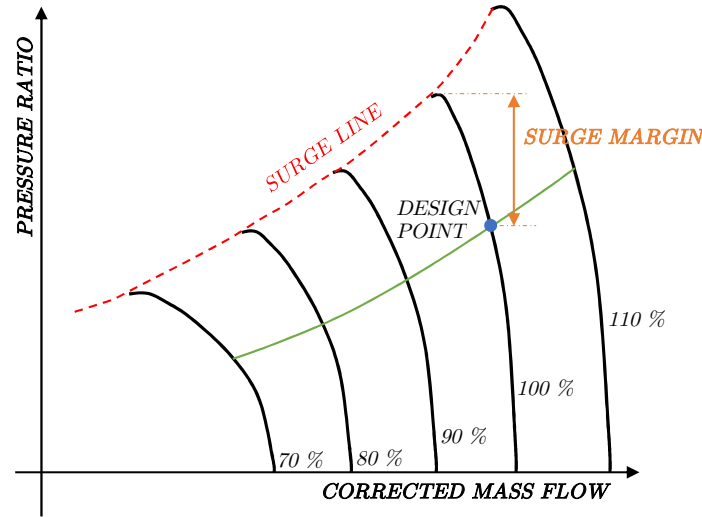


Fig. 1.2: Example of a typical compressor map.

The presence of a surge margin ensures that the engine’s operating condition is always far from unstable and small variations are not likely to cause a compressor stall. Despite the margin, an engine can still suffer a compressor stall for a number of causes. These can be divided into 4 categories:

Damage to the blade - Damage to the blade airfoils reduces their capability to impart turning to the flow, in particular it reduces the maximum value before separation occurs. As a result, the surge line moves downwards reducing the range of stable operating conditions and possibly triggering a stall. The damage to the airfoil can be caused by the ingestion of foreign objects like debris or by a bird strike. The rubbing of blade tips against the casing caused by fast acceleration or thermal expansion damages the blade tip; if the distance between the rotor blade and the casing increases, the blade becomes more susceptible to separation and the surge margin reduces. The ingestion of sand and dirt for a prolonged time can cause erosion of the blades or, at high temperatures, foreign matter can build up on the airfoils and change their shape, an effect called *degradation* [4].

VSV *mal*schedule - The VSV (Variable Stator Vanes) are variable geometry blade rows used to reduce the incidence angles on the blades when the compressor operates at low mass-flow; practically they increase the surge margin over a range of operating conditions. However if a *mal*-schedule occurs they can be stuck at the low speed stop (LSS) or at the high speed stop (HSS), become mismatched and reduce the surge margin.

Fast transients - During fast transient manoeuvres the operating point leaves temporarily the equilibrium line and moves closer to the surge line. A typical example is a *slam* acceleration: when the engine is accelerated, the operating point moves closer to the surge line. The faster the acceleration is, the closer to the surge line the operating point will pass. In a typical 3-spool engine architecture a similar behaviour can occur also during fast deceleration; in this case the operating point of the IPC rises above the equilibrium line and moves closer to surge line [10]. On modern engines the EEC (Electronic Engine Control) limits the rapidity at which the fuel is changed, to avoid this type of occurrences.

Distortion of the inlet flow - A perturbation of the inlet flow can cause the compressor to work in less favourable conditions and cause a compressor stall.

During extreme flight manoeuvring or strong side-winds the flow can separate in the intake, causing areas of the compressor to receive air at very low total pressure and increasing the loading of the airfoils. A similar increase in loading is obtained if hot gases are ingested, this can occur because of the operation of a thrust reverser (the reversed exhaust is re-ingested by the engine) or when a fighter jet fires a missile. Finally the ingestion of ice can cause a distortion as the solid ice particles will disturb the airflow in the blade rows. This normally happens when ice forms at the inlet of the engine core and is then shed and ingested by the compressor.

Any combination of these factors must also be taken into account. For example a small blade damage might not be enough to cause a stall by itself but it would lower the surge margin and therefore the capability of the compressor to resist inlet distortions. To ensure that even a combination of factors will not cause instability, a sufficient surge margin must be achieved during the design of a new engine. The surge margin chosen must be further increased as part of it will be lost in time due to degradation caused by sand and dirt. At the expected engine end-of-life the amount of margin left must still be sufficient to guarantee safe operation. Paduano [5] reported an example of the estimation of this surge margin from the “ARP 1420 Aerospace Recommended Practice: Gas turbine engine inlet flow distortion guidelines”, table 1.1. From this table it can be seen that up to 25% of the pressure ratio is given up to maintain a good surge margin; a similar estimation is reported also by Walsh and Fletcher [6].

Despite many years of research current tools are still insufficient to model and predict compressor stall. In his 2016 review of the last 75 years of research Day states that although we can now explain what happens during compressor stall “we have little to offer in the way of rigorous rules of designing a more stable compressor” and “we can seldom predict what will happen before we start” [2]. The little predictive capability and the necessity of leaving a large surge margin, forces the designer to reduce the load on each stage; this in turn results in heavier and bulkier compressors. Commercial CFD codes can be used to model an unstable compressor, however they require computational resources so large that they can not be used practically. Other

Destabilizing effects	Fan stall margin budget	
OPERATING LINE	Random	Non Random
Inlet Distortion	0.5	-
PLA Transients	4.0	-
Variable Geometry Control Tolerances	-	± 1.1
Engine to engine variation	-	± 1.2
SURGE LINE REDUCTION		
Reynolds number	1.87	-
Inlet distortion	6.25	-
Engine to engine variation	-	± 1.3
TOTAL MARGIN REDUCTION	12.62	± 2.083
BASE SURGE MARGIN	25.00	
NET SURGE MARGIN	10.30	

Tab. 1.1: Typical stall margin budget, from Aerospace Recommended Practice ARP 1420, reported by Paduano [5]. All margins expressed as percentage of the operating line maximum pressure ratio.

methods with lower computational costs exists but they struggle to capture all the features of compressor stall and have never reached a level of maturity sufficient to be applied for commercial use. A review of these methods is reported in chapter 3.

Therefore, there is a need for a tool capable of simulating compressor stall within a reasonable computational time, predicting which scenario will occur, and assessing how design choices can affect the stability and the post-stall performance of the compressor.

The chaotic event of compressor stall can take multiple forms, but these are usually divided in two main types: *rotating stall* and *surge*. These are described in detail in section 3.1.2. In the following thesis the capability of modelling compressor stall is also referred to as the capability of modelling these two events.

1.2 Shaft failure and turbine over-speed

A shaft failure followed by turbine over-speed is one of the most critical and destructive events a jet engine can suffer. Shaft failure occurs when one of the shafts connecting the compressor and the turbine breaks and stops the power transfer between the turbo-components. Such event can occur both as a primary failure (because of shaft fatigue, corrosion or defect) or as a consequence of another component's failure like an oil fire or a bearing breakage. Immediately after the shaft breaks the flow conditions remain substantially the same and the turbine continues to extract the same power from the air. This power however is not transferred to the compressor because of the broken shaft and instead accelerates the turbine. The accelerating turbine reaches rapidly a maximum speed, called *terminal* speed. The typical time of this over-speed acceleration varies from hundreds of milliseconds up to seconds depending on the operating conditions. Depending on the sizing of the rotor disc, before reaching *terminal* speed there can be another speed at which the centrifugal forces are sufficient to break the disk, this is called *critical* speed. If the turbine disc bursts, it separates in high energy fragments which can leave the engine (uncontained failure). The fragments have such high speed and energy that after escaping the engine they can still cut through the fuselage or through other engines. The fragments pose a direct threat to the passengers and crew if they reach the cabin but they can also damage other vital systems of the aircraft.

The certification Authority requires for an aero-engine to be certified that the turbine rotors must be able to reach specific speeds without bursting. According to EASA 'CS-E 840 Rotor Integrity' [7] the turbine rotors must be able to resist for 150 seconds at 120% of the maximum speed permissible during normal operations or at 105% of the maximum speed which can occur because of the failure of a component/system. The maximum speed caused by shaft failure (the *terminal* speed) is normally the highest of all the speeds caused by component failure and it constraints the design of the rotors causing a large over-sizing.

It is possible not to include the maximum speed resulting from shaft failure in the rotor certification (and therefore reduce the over-sizing) if it can be demonstrated

that this over-speed occurs at a rate less than *extremely remote*¹ (according to the EASA ‘CS-E-850 Compressor, fan and turbine shafts’, [7]). The shaft failure event itself can be *remote*² as long as the engine can safely be shut down without any disc burst.

To comply with this requirement, engine manufacturers have introduced safety systems to avoid uncontrolled turbine over-speed and reduce its occurrence to extremely remote [8]. All the shafts (LP, HP and where present IP) can suffer a shaft failure however each turbine can be slower or faster in reaching the speed at which it burst. While this can depend on many design choices, in general HP and IP turbines reach *critical* speed faster because of their lower inertia. The main system to prevent turbine over-speed is the immediate shut-off of the fuel flow and VSV closure to reduce the power supplied to the turbine by the airflow. This safety system is activated by the EEC (Electronic Engine Control, sometimes known as FADEC, Full Authority Digital Engine Control) which relies on shaft speed monitoring to detect an over-speed. Unfortunately the probes measuring rotational speed are normally located in the front part of the shaft and when the shaft breaks they can only see the front part of the shaft decelerating and can not detect the turbine over-speeding. While on the LP shaft the probes can be positioned at both ends, the high temperatures present at the back of the IP and HP shafts make it very challenging to have a measurement there. Other strategies exist to prevent over-speed which do not rely on speed monitoring but on events which occur automatically after a shaft failure. These events are:

- As the shaft breaks the turbine rotor (or rotors) becomes dislocated axially and is pushed backwards by the airflow against the stator. If the two blade rows get close enough to interact, the blades will be destroyed and the turbine will not be able to extract any more power; this effect is called *blade tangling*.
- As the rotor disc is pushed backwards it will rub against the hub of the stator blade row and cause friction, which will slow down the rotor. The amount of

¹ An extremely remote even has a probability lower than 10^{-7} of occurring during the aircraft operational life.

² A remote event has a probability lower than 10^{-5} of occurring during the aircraft operational life.

friction depends on the shape of the two structures. As the rubbing will cause the metal to melt away, the shape of the structures in contact and the friction produced will change in time.

- The shaft failure is very likely to start a compressor stall, this will decrease the amount of pressurised air supplied to the turbines and possibly prevent the over-speed.

The occurrence and magnitude of these events are very difficult to predict and control and thus it is not easy to make them a reliable safety system. Which exact strategy is employed by each engine manufacturer to prevent turbine over-speed is proprietary information and not described in this thesis.

In case of a shaft failure the compressor (or compressors) are likely to suffer a stall because of two main reasons [8]:

- Depending on the position where the shaft breaks, the compressor can disengage from the rear bearings and start vibrating. The vibrations cause the tip clearances to change: some blades will move away from the casing increasing the clearances while others will rub against the casing. The increase in tip clearance reduces the surge margin and a compressor stall is likely to be triggered.
- As one of the compressors disengages from the turbine and decelerates, the whole compression system (HPC and IPC/booster) becomes mis-matched. As a result one of the compressor's operating point can cross the surge line and trigger a compressor stall.

The presence of a compressor stall during the shaft failure event is often critical in determining whether a disc burst will occur. However during engine testing the compressor has been observed experiencing different combinations of the above effects, for example vibrations occurring and then stopping or compressor stalling and then quickly recovering. Without a proper capability to model a compressor stall these macroscopic effects seem to occur randomly. Therefore, the correct modelling of compressor stall is essential to any attempt of predicting the shaft failure outcome.

In 2007 at Cranfield University a project was started with the objective of attaining the capability of modelling shaft failure. PhD projects from Gallar [9], Soria [8] and Pawsey [10] spanning from 2007 to 2017 have focused on the creation of WEST (Whole Engine Simulation Tool), a tool to model the whole engine during the shaft failure and turbine over-speed event. This tool is described in section 3.2.6. The tool requires as an input the performance of the compressor during stall, in particular during reverse flow and during rotating stall, described by means of characteristic line. These characteristic lines are described in section 3.1.3.

The necessity to correctly model the compressor stall has prompted the start of a parallel project focusing on this subject. This PhD project started in 2010 with Zoppellari [11] who created a two-dimensional tool capable to model compressor stall in low-speed compressors, this is reported in detail in section 3.2. The success of her PhD led Cranfield to continue the study on compressor stall modelling with the current PhD project.

1.3 Project aim and objectives

The main aim of the project here presented is to support the shaft failure and over-speed studies within the Cranfield UTC by developing a tool capable of predicting post-stall compressor performance, which is a necessary input to the code WEST. The tool must be able to model rotating stall and surge in 3D and predict the rotating stall and reverse characteristics of a multi-stage axial compressor. The computational time required must be much smaller than what is currently necessary, for the same application, using three-dimensional high-fidelity commercial CFD.

The objectives of this doctoral project have been identified as:

1. The creation of a fast, robust and reliable 3D through-flow code based only on compressor geometry and first-principles, physics based correlations. The code must be able to simulate multiple cases of stall at different operating conditions to produce the compressor characteristics. It must have sufficient resolution in space and time to satisfactorily represent the modes and structures of the flow-field during stall inception and development while keeping the computing

time within a realistic order of magnitude.

2. The validation of the solver, the body force method and the forward flow empirical correlations by matching the results for steady state, unstalled performance to experimental data.
3. The validation of the reverse/separated flow treatment by matching of the reverse flow characteristic to the available experimental data.
4. The validation of the rotating stall and surge simulation capability with the available data from low-speed experimental rigs. The accuracy in predicting rotating stall characteristics, stall cell structures and stall/surge dynamics will be assessed.

1.4 Thesis outline

The background to this research was discussed in chapter 1. The initial problem has been outlined as the lack of a fast and reliable method to model compressor stall. Benefits of developing a new tool have been found in the fields of compressor design, engine operation and turbine over-speed protection. The objectives of this PhD have been identified and described.

In chapter 2 a brief overview of the project is presented. The contribution to knowledge of the project and its industrial relevance are also discussed.

Chapter 3 provides an extensive literature review on three main topics essential to this project. The first part reports the theory about compressor stall, detailing the current understanding of this complex phenomena. Stall inception, dynamics, flow-field structures, performance and characterization are discussed. The second part of this chapter reviews the types of code existing to model compressor stall. For some meaningful cases a more detailed description of the methodology is reported. The last section describes how compressor stall is tested experimentally. By means of some key examples, the details of typical compressor rigs are discussed: which measurements are available, how they are reported, how the operating conditions

are simulated. Data from the rigs described are also used for the validation of the code in chapter 6.

Chapter 4 illustrates the methodology adopted in the development of the code. Three main sections describe the solver, the body force method, and the empirical correlations. The boundary conditions, the input and output of the code, the method in which the code is used and the programming are also discussed.

Chapter 5 reports the cases which have been modelled to validate the new tool. The availability of original geometries and measurements is described and any assumption taken is discussed.

Chapter 6 reports the results obtained and the validation of the code by comparison with experimental data. The chapter follows the steps of validation. First the Euler solver is verified modelling the flow in an empty duct, then the unstalled forward flow is validated on a 2 stage compressor case, then the reverse flow on 2 multi-stage compressors. The rotating stall is simulated and validated on two low-speed rigs, the structure of the flow during stall is also investigated in details and stall cell size, shape and speed are matched to experimental data. Finally surge is modelled and compared to experimental data, and a parametric study is carried out to assess how the model responds to different tank volumes and inlet distortions. In the last section a grid convergence study is also undertaken and discussed, to highlight the uncertainty in the results produced due to domain discretization.

Chapter 7 summarises the results obtained, the validations achieved, and the contributions to knowledge introduced by this project. It also contains the author's recommendations for the future development of the code.

Chapter 2

Project overview

The current PhD project has the objective of developing and validating a new methodology for the modelling of compressor stall. From this methodology, a new tool is created capable of simulating a compressor during reverse flow, rotating stall and surge. The tool development has been divided into 3 stages which correspond to 3 successive validation processes. For every validation stage the code requires specific capabilities, listed in figure 2.1.

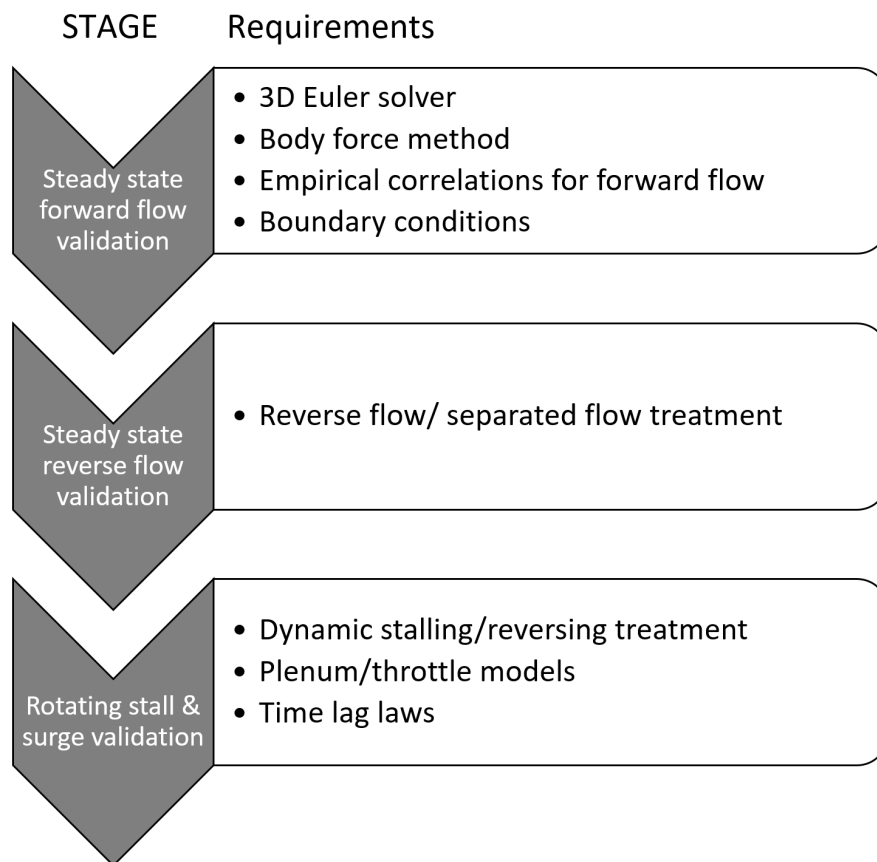


Fig. 2.1: Project roadmap.

2.1 Contribution to knowledge

Only one validated through-flow code exists in literature which can simulate 3D rotating stall, however it requires a complex scheme which solves the Euler equations plus an additional equation. The details of this code are not in the public domain and the method used can not be easily reproduced. The only other validated alternative to model compressor stall remains 3D CFD, but because of the computational cost it is not a viable option for effective use. The purpose of the current project is in fact the creation of a new 3D through-flow code capable to model all types of compressor stall. In this thesis the new code is presented together with its validation.

The solver used for the new code is the Godunov scheme. Despite being only a first order solver it is a monotone code, which guarantees an improved robustness when compared to classic FVMs (finite volume method). It also produces a physically correct modelling of the pressure waves, which play an important role in the transient dynamics. At the best of the author's knowledge this method has never been used in a 3D through-flow code applied to the modelling of compressors' post-stall transient behaviour. The features of the solver are deemed particularly beneficial for this type of application. The implementation and extensive testing of this new solver are the direct outcomes of this work, with simulated results comparing very favourably against experimental data.

The treatment of separated flow in blade passages and the body force modelling of blade rows during flow reversal is a very complex topic. Many body force formulations can be found in literature, with more or less physical background and different levels of complexity. Only few of them have been successfully used for 3D reverse flow modelling. In this project a new physics-based formulation is proposed which is general enough to be applied to different compressors without extensive re-calibration, yet it is simple and easy to implement.

2.2 Industrial relevance

Once completed, the tool proposed in this PhD can be used to support the design of new compressor for commercial use. In particular it can give the capability of:

- predicting the rotating stall and reverse flow characteristics of an axial flow multi-stage compressor,
- modelling in detail the flow-field during compressor rotating stall and surge,
- understanding the effect that compressor design, overall engine geometry and operating point have on stall triggering and development,
- predicting the dynamics of rotating stall and surge and the pressure loading that the engine structure is subjected to during these events.

All of these capabilities are not available to designers without either using extensive computational resources for CFD or expensive experimental testing. If such a tool were available during a new engine design it could be used to

- improve the normal operability of the compression system,
- modify the compressor geometry to avoid compressor stall or reduce its severity,
- enhance the recoverability,
- support research on techniques to control/prevent stall.

Furthermore the maximum pressure loading during surge is one of the limiting factors during the mechanical design of compressor blades, core ducting and fan ducting. This loading is unknown when a new product is designed and large safety factors are required to take it into account. Thanks to the low computational cost required to model a compressor stall with the method proposed, the new tool could be used to simulate many surge cycles and achieving a better estimation of these mechanical loadings. A better estimation of the loadings would allow a more efficient design of some large and heavy engine components.

Chapter 3

Literature review

This literature review has been structured in three parts. The first part discusses the theory of compressor stall, its inception, dynamics and characteristics. The second part is a review of methods to model compressor stall. The objective of this review is to present the state of the art of compressor stall modelling as well as identifying the pros and cons of different techniques. The knowledge gathered with this review is used to support the choices made in developing the current methodology. The third part describes the experimental techniques employed in testing compressor stall on real rigs. A couple of meaningful cases are presented; from these rigs comes the experimental data used to validate the present model. The objective of this section is to familiarise the reader with the operation of a compressor rig, the experimental measurements available and the post-processing method used to quantify/visualise the compressor stall performance and flow-field.

3.1 Compressor stall theory

This section contains a description of the current knowledge and theory on compressor stall. Hereafter the expression *compressor stall* will be used to indicate any of the possible behaviours of a compressor after the stability line is crossed and the compressor is not in a forward unstalled flow regime.

3.1.1 *Stall inception*

The expression *stall inception* refers to the initial growth of perturbations in the compressor flow-field which immediately precedes stall. Stall inception has been

largely studied and two main types have been identified, the *modal* and *spike* inception [12, 13].

The typical situation in which stall occurs is when the operating point of the compressor moves along the characteristic speed-line towards the maximum pressure rise. When low-speed experimental rigs are tested the maximum pressure rise occurs when the characteristic becomes horizontal. The compressor is stable only when operating on the negative sloped side of the pressure rise characteristic; when the operating point crosses the horizontal part of the characteristic it becomes unstable. Depending on the operating condition the various stages of a compressor can be at different positions on their individual stage pressure rise characteristics (figure 3.1). Some of them can be in an unstable condition while the compressor as a whole is still stable.

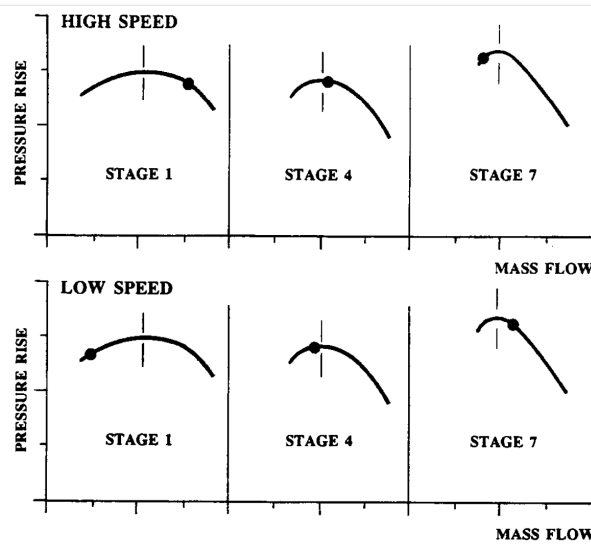


Fig. 3.1: Stage total-to-static pressure rise characteristic of a multi-stage compressor at different operating conditions [14].

When the whole compressor operating point reaches the horizontal part of the characteristic line a 2D perturbation in velocity appears across all of the stages. This perturbation has a sinusoidal shape in the circumferential direction and rotates around the annulus at a fraction of the rotor speed. Depending on the operating conditions this perturbation can reduce and disappear or grow into a rotating stall with a smooth transition. The second case is known as the *modal* inception. When modal stall inception is studied experimentally and the flow coefficient is probed at

various circumferential positions, the inception can be observed in the traces tens or hundreds of revolutions before the stall actually occurs.

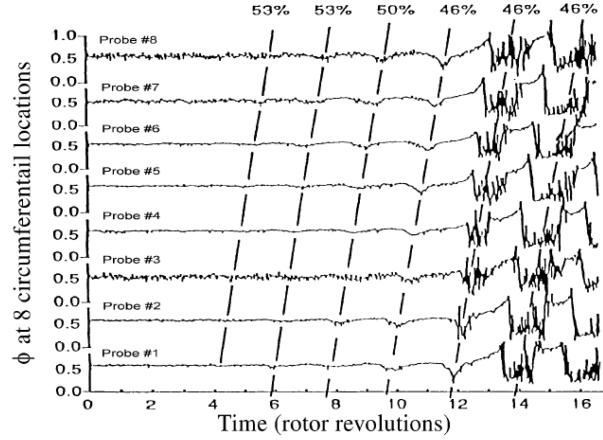


Fig. 3.2: Waves from a modal stall inception visible in measurements of axial flow coefficient at various circumferential positions [11].

Depending on the compressor geometry it is possible that, while the operating point is still on the stable side of the characteristic, in one of the blade passages the incidence angle will exceed the critical value. In this case, the flow in that specific blade passage will separate. After the first blade row separates, an area of reduced velocity will form suddenly and initially cover only few blade passages. This area, called *stall cell*, grows very fast and covers the whole annulus in usually less than five revolutions from inception. The velocity at which the growing stall cell rotates is around 70-80% of the speed of the rotor itself. As both Day [15] and Jackson [16] observed, the starting point on the cell appears to occur always on the same blade. Day even reported it occurring when the same rotor blade was in a particular position with respect to the stator. This finding suggests that as the compressor approaches stall there is a specific blade with an imperceptible defect which will start a localised cell. This type of inception, unlike the modal one, is irreversible and once appeared it will always lead to a flow breakdown.

This initial perturbation leaves a spike on the measured flow coefficient traces and is known as *spike* inception. In modern high-speed compressors stall inceptions are usually of the spike type.

In spike type inceptions the initial separation often occurs close to the rotor tips and in the past it has been linked to the interaction with the back-flow from the

clearances. However it has later been proved that it occurs close to the tip even with shrouded compressors, where no back-flow exists [17]. The Work by McDougall et al. [18] suggests that the position of the initial stall is mainly driven by the blockage.

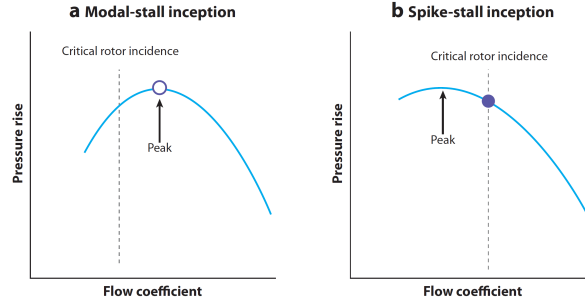


Fig. 3.3: Different operating point at which modal and spike inception occur [12].

The inception mechanism described here occurs if a compressor is slowly moved towards the surge margin by mean of throttle closure, VSV malfunction or fuel spike. There are of course other triggers which can cause an immediate stall even when far away from surge limit. There are for example: inlet distortion, foreign object damage, blade loss, vibrations, etc.

3.1.2 Compressor stall types

Following the stall inception the operating condition will change rapidly as the compressor enters the post-stall transient. The compressor will follow a specific trajectory on the compressor map and most likely settle into one of two main types of periodic instabilities: Rotating stall or Surge.

Rotating stall

Rotating stall is characterised by one or more regions of stalled flow called *cells*, which rotates around the annulus at a fraction of the rotor speed. Rotating stall can be observed during most of the transient conditions which occur during any compressor stall. The expressions *mature* or *fully developed* rotating stall instead are used to indicate the situation in which the compressor exhibits a steady rotating stall with a constant operating condition. In this section is described the mature rotating stall.

In a mature rotating stall, the stalled and unstalled sectors are clearly defined and sharp discontinuities divide them. In multi-stage compressors the cell can span axially across all of the stages, or be confined to the front ones. In the first case, the cell often covers the full blade span and only one large stall cell is present. Instead, when the rotating stall is present only in the first stages there are normally multiple cells which cover only part of the span; this has been reported experimentally by Dodds [19] and Day [14], [20].

The stall cell rotates around the annulus always in the same direction of the rotor but at a slower speed, in a range between 20% to 60%. Normally part span cells rotate faster than full span ones. The shape of a fully developed cell has been reported by Jackson [16] which inferred it by the measurement of hot wires. Figure 3.4 illustrates the stall cell at different moments during its development into a mature stall; by its 5th appearance it had reached a steady condition.

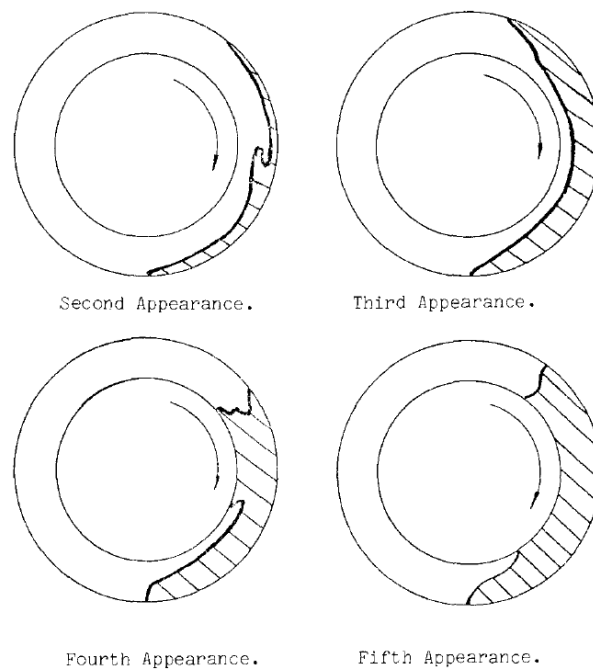


Fig. 3.4: Stall cell developing into a mature rotating stall [16].

Although a fully developed rotating stall is not normally able to directly damage the compressor, it greatly increases its exit temperature because of the reduced efficiency. As a result the stall must be quickly interrupted to avoid over-heating of the turbine. Unfortunately in some engine architectures rotating stall is very stable

and difficult to recover from. In this case the only option to avoid damage is to reduce the fuel flow or shut down the engine.

The current description of the stall cell structure has been introduced by Day and Cumpsty [20]. It had been previously proposed that the stall cell was a dead wake like the one created by a bluff body trailing across the stage and not exchanging mass with the unstalled flow. This is known as the passive stall cell theory. The current theory, proposed by Day and Cumpsty in 1978, is called *active* stall cell theory and considers the stall cell as a region of highly energetic flow which continuously exchanges mass-flow with the unstalled region through its lateral borders. The active cell does not follow the flow direction but is mostly directed axially and does not spiral around the annulus as a dead wake would do. The difference in the two models is shown in figure 3.5.

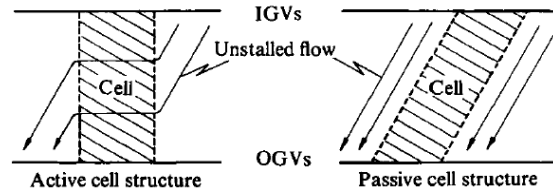


Fig. 3.5: Passive and active stall cell models [20].

The same authors also provided a description of the flow inside of the cell and how it is influenced by the compressor geometry. When a multi-stage compressor with a low design flow coefficient ϕ^* was forced into rotating stall, the stall cell observed did not exhibit any reverse flow but only a reduction in velocity. When the compressor was re-staggered for higher design flow coefficient, the stall cell it developed had some areas showing reverse flow. The ratio of reverse flow to forward flow increased with the ϕ^* , see figure 3.6. The presence of reverse flow in the cell produces a high swirl flow appearing in front of the rotor tips which can reach velocities higher than the blade speed U .

The first description of the mechanism which determines the stall cell propagation has been given by Emmons [21] and is presented in figure 3.7. According to this theory the reduced velocity of the stalled passage influences the upstream flow-field modifying the incidence angle on the adjacent blades. The forced stalling and un-

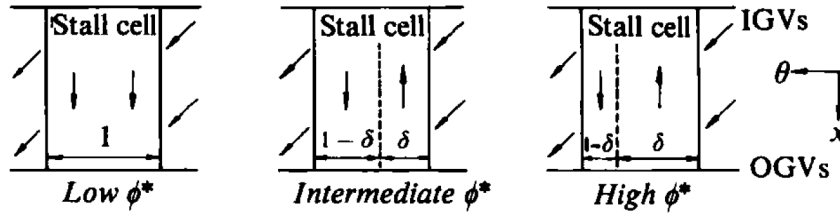


Fig. 3.6: Effect of the compressor geometry on the stall cell structure [20].

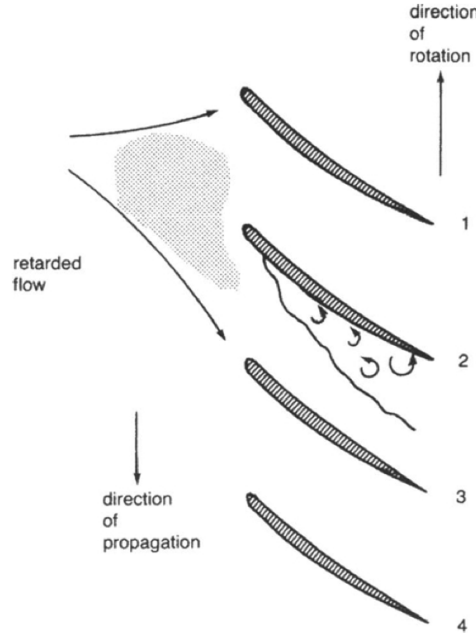


Fig. 3.7: Stall cell propagation mechanism as proposed by Emmons [21].

stalling of the neighbour blade passages determines the cell propagation. However in the case in which the flow is reversed inside of the stall cell this propagation mechanism would be impossible. Day and Cumpsty [20] then suggested that the propagation is dictated by the blade rows acting like paddles moving around the stalled flow. This theory was then further developed by Cumpsty and Greitzer in 1982 [22] who proposed as the main mechanism of cell propagation the inertia of the flow inside of rotors and stators in adapting to the pressure perturbations. The matching of these inertias in the relative and absolute frame of reference sets the propagation speed of the cell. According to Longley [23] the inertia matching is always the main drive of the cell propagation, however the perturbation of incidence angle (i.e. the mechanism proposed by Emmons) and the viscous effects can affect the propagation speed. The importance of these contributions depends on the blade

design, and was observed only in the small-amplitude case Longley modelled, where no reverse flow is present. The mechanism described by Emmons has also been suggested to be the mechanism which propagates the initial cell generated during a spike inception [17]. This type of cell rotates much faster than a mature one, which might depend on the different mechanism of propagation.

Surge

Surge is a cyclic stall in which the compressor experiences periodic deceleration and acceleration of the flow, with the compressor operating unstalled for part of each cycle. The cycle can be divided in four parts, starting from stall inception:

1. As the compressor departs from the stability line it normally exhibits a rapidly growing rotating stall and the pressure ratio produced drops dramatically, typically $\sim 50\%$ of the initial pressure ratio. As a result, there is a strong reduction in mass-flow in the whole compressor. After a few revolutions the stall cell covers the whole annulus and the flow becomes mostly axi-symmetric. The overall mass-flow can even become negative, with the flow exiting from the compressor inlet.
2. As the mass-flow delivery reduces, or inverts, the volume behind the compressor will de-pressurise and the pressure ratio across the compressor will fall.
3. When the pressure ratio reduces under a certain value, the compressor is able to re-accelerate again the flow in the downstream direction. As the flow re-accelerates the blade rows gradually unstall.
4. Once the compressor is fully unstalled, the operating point is again on the original speed-line at a low pressure ratio. The downstream volume pressurises again and the operating point moves towards the surge limit. Unless the operating conditions have changed since the last cycle the compressor will stall again and start another cycle.

The downstream volume considered in this description is in fact either the combustion chamber in a real engine or the tank in an experimental rig (see section

3.3).

The 4 parts of the cycle can be clearly identified in the trajectory found experimentally by Day [58] in figure 3.8.

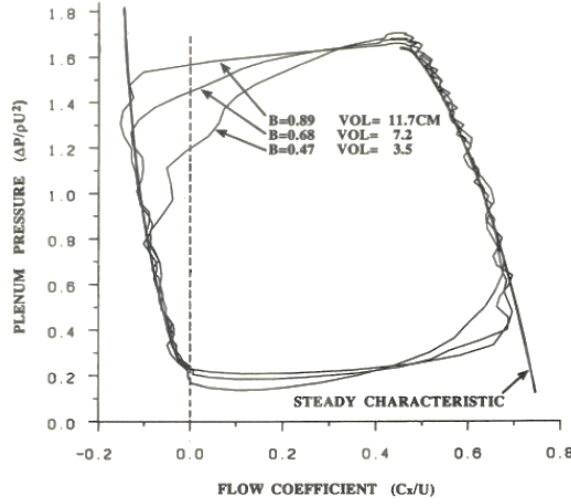


Fig. 3.8: Surge cycle trajectories for different values of downstream volume [58].

If during the cycle the mass-flow (or the axial flow coefficient) goes below zero the stall is called *deep surge*, if it does not it is called *classic surge*. It is important to notice that even if in classic surge the total mass-flow remains positive, the flow can reverse locally.

During the reverse and unstalled parts of the surge cycle the flow remains mostly axi-symmetric. The reversal of the flow is started by a transient rotating stall which grows to cover the whole annulus [58]. According to the author of this thesis the rapidly growing stall cell observed should be considered a spike inception evolving rather than an actual rotating stall, although the difference is subtle. A similar behaviour was reported also during the beginning of the recovery; in this case bubbles of unstalled flow will form and grow until they cover the whole annulus [57]. According to experimental finding, this recovery process takes less than half the time than its stalling counterpart [58].

The frequency of the cycle is experimentally reported to be in the range of 3-10 Hz on jet engines. However, it can vary greatly depending on the volume downstream of the compressor; in particular, bigger volumes produce lower frequencies and smaller volumes produce higher ones. Surge frequency was previously thought to coincide

with the Helmholtz frequency of the compressor volume but it was later proved to be wrong [58].

The violent pulsations in mass-flow rate occurring during surge are responsible for the loud bangs heard during compressor stall. The strong pulsations in pressure and vibrations can damage the blade rows if the surge persist for several cycles. Continuous surge can even destroy the engines, an example is the accident of Scandinavian Airlines Flight 751 where the pilots failed to recover from surge for several seconds and both the engines became unusable [24].

3.1.3 Stalled performance characterization

A complete compressor map is a map which contains characteristics for forward flow, reverse flow and rotating stall. These maps are normally pressure rise coefficient ψ over flow coefficient ϕ and are valid for a single rotational speed. Only steady operating conditions can be used to create a characteristic, thus only mature (steady) rotating stall and reverse flow are shown on the map. An example is shown in figure 3.9.

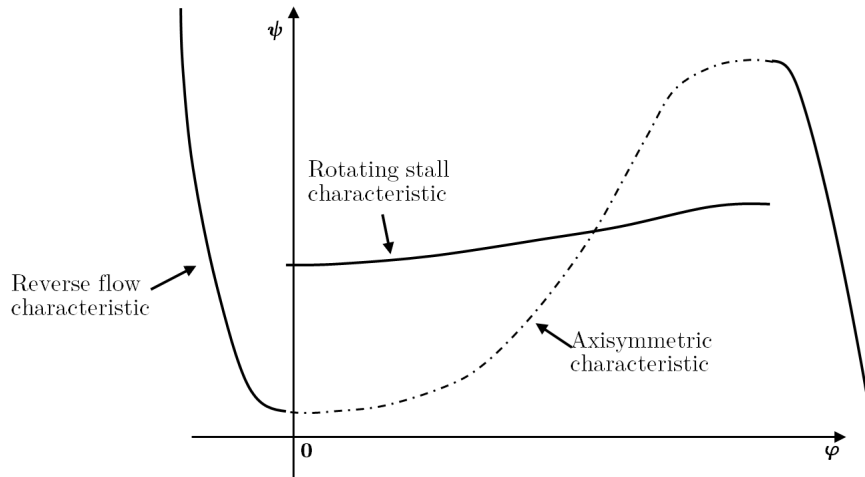


Fig. 3.9: Example of complete compressor map.

Steady reverse flow can be obtained experimentally by imposing a constant pressure gradient across the compressor. Putting together the operating points at different pressure ratios a characteristic can be drawn. The reverse flow characteristic has normally a steep slope and intercepts the zero flow coefficient line at a value of

$\psi > 0$. This is because the rotating blade rows always produce on the flow some axial force directed downstream and there must be a balancing pressure gradient for the overall mass-flow to be steady at 0. The point at $\phi = 0$ is not usually part of the steady characteristics as the compressor starts the recovery before reaching it.

The rotating stall characteristics is the locus of all the operating points of *mature* rotating stall. It is usually a quite flat line and partly overlaps the forward flow characteristic, thus creating some hysteresis. Gamache [57] reported some hysteresis occurring also between the rotating stall and reverse characteristics.

An axi-symmetric characteristic can also be drawn, though it is not a *steady* condition. This represents the operating condition of a stalled compressor which has not developed any rotating stall. This was initially introduced by Moore [25, 26, 27], however since a stalled compressor always develops a rotating stall in few revolutions it is impossible to observe such condition experimentally in steady-state. Proof of its existence has been given experimentally by Longley and Hynes [28]. They restaggered the rear stages of an experimental compressor to make them less prone to stall. They were then able to run the first stage in an axi-symmetric stalled condition while the rear blade rows maintained them stable. Koff and Greitzer developed a technique to draw the axi-symmetric line based on the transient performance during surge. However, the characteristic can be also obtained by simply fitting a cubic spline to the unstalled and reverse characteristic extremes and this method has been successfully used to model surge analytically [2].

The connection between the axi-symmetric and rotating stall characteristics has been suggested by Paduano [5]. He used a two-dimensional lumped code to simulate both surge and rotating stall. Figure 3.10 shows the characteristics used and the simulated stall trajectories for different values of downstream volume (constant throttle and speed). Depending on the downstream volume the outcome is either surge or rotating stall, however no rotating stall characteristic is input in the code. In this 2D code (having axial and circumferential discretization) the pressure rise across the compressor at a specific circumferential position is based on the local flow coefficient. The flow coefficient can be reversed, stalled and unstalled at different θ positions of the same compressor, and the appropriate part of the characteristic

is used in each of them. That is, the 2 sectors are locally operating as if in axisymmetric conditions. The rotating stall performance is then obtained naturally as a combination of the others. This is further supported by the clear distinction of the stalled and unstalled sectors during rotating stall.

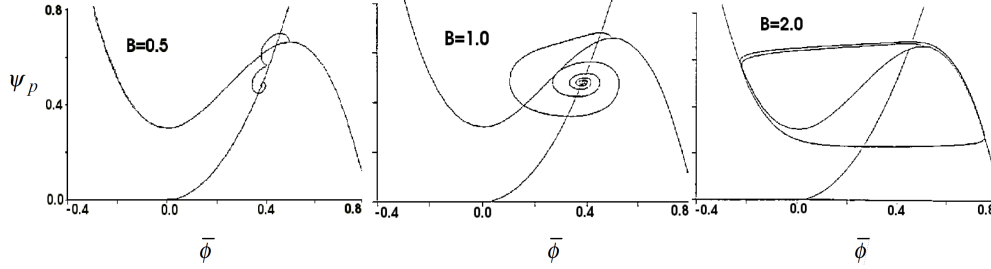


Fig. 3.10: Stall trajectory simulated with a 2D lumped model, with different downstream volumes [5].

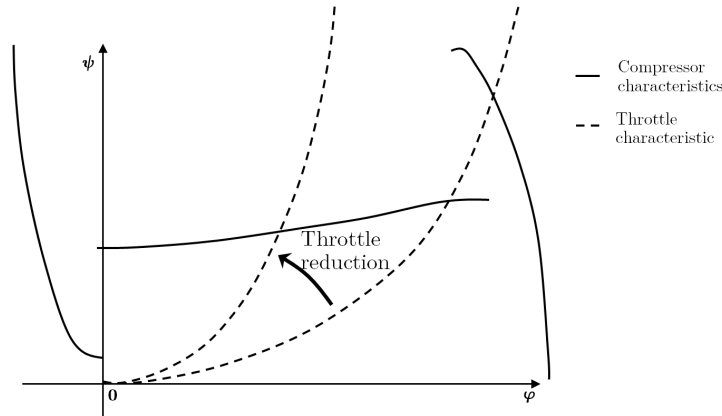


Fig. 3.11: Complete compressor map with throttle characteristic.

When testing an experimental rig, a plenum can be set downstream of the compressor, exhausting to the ambient pressure through a throttle. A throttle characteristic can be introduced to describe the effect of the throttle area on the operating point. This characteristic links the mass-flow exhausted through the throttle with the pressure jump between the plenum and the atmosphere. If a plenum and throttle is employed, a steady operating condition can only be reached at the intersection of the throttle characteristic and the rotating stall or unstalled characteristic. From image 3.11 it is easy to understand how, by closing the throttle, the compressor can be forced into stall. Also it can be observed that a hysteresis exists between the unstalled and rotating stall line (see also figure 3.18). Unlike the axisymmetric

characteristic, the throttle line starts from the $[0, 0]$ point of the map.

In real engines a link exists between the pressure in the combustion chamber and the exhaust through the NGVs and the nozzle. However, the mass-flow also depends on the temperature in the combustor and on the secondary air system, and it is not so straight-forward to draw a characteristic line equivalent to the throttle characteristic.

3.1.4 Compressor stall dynamics and stability

The dynamics of stall play an important role in determining which type will occur: rotating stall, classic surge or deep surge. For example, during a surge cycle the pressure at the back of the compressor is determined by the downstream volume and how quickly it will depressurise and repressurise. Other factors influencing dynamics are the compressor performance (function of geometry and rotational speed), the ducting and the throttle capacity. This is true for compressor rigs only, even more factors must be included when considering real engines.

In compressor rigs it is easy to modify the volume of the downstream plenum and assess its influence on the compressor's stability. It has been observed many times that as the volume is reduced the compressor behaviour will change from exhibiting deep surge to getting locked in rotating stall [58, 57, 2].

With very large volumes the back pressure changes much slower than the time required for the flow to invert direction. In this case the surging compressor reverses and recovers at almost constant pressure ratio and the operating point follows closely the reverse characteristic during the blow-down. If the volume is reduced the surge trajectory shows more of the compressor dynamics. The operating point overshoots the peak pressure ratio when stalling. The reversing and recovery parts of the cycle also become oblique, as in fact the pressure in the volume will change substantially during them. If the volume is reduced even further during the reversing of the flow the pressure will adjust so fast that the compressor will recover before reaching the zero mass-flow line resulting in a classic surge. All these trajectories are illustrated in figure 3.12. As the volume is reduced further, the re-pressurization of the plenum becomes so fast that the compressor is unable to reach the unstalled line again.

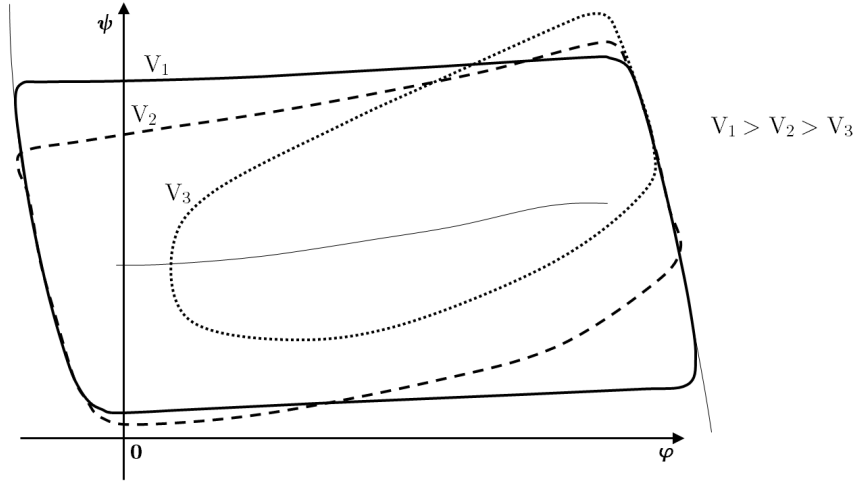


Fig. 3.12: Surge trajectory for different volumes.

In this case, the oscillation reduces and following a spiral trajectory the operating point finds an equilibrium on the rotating stall characteristic. The equilibrium point is at the crossing of the throttle and the rotating stall characteristic. The operating point does not arrive on the axi-symmetric characteristic because in the meanwhile a rotating stall has developed. For some geometry or operating condition the compressor post-stall behaviour might change from deep surge to rotating stall without classic surge.

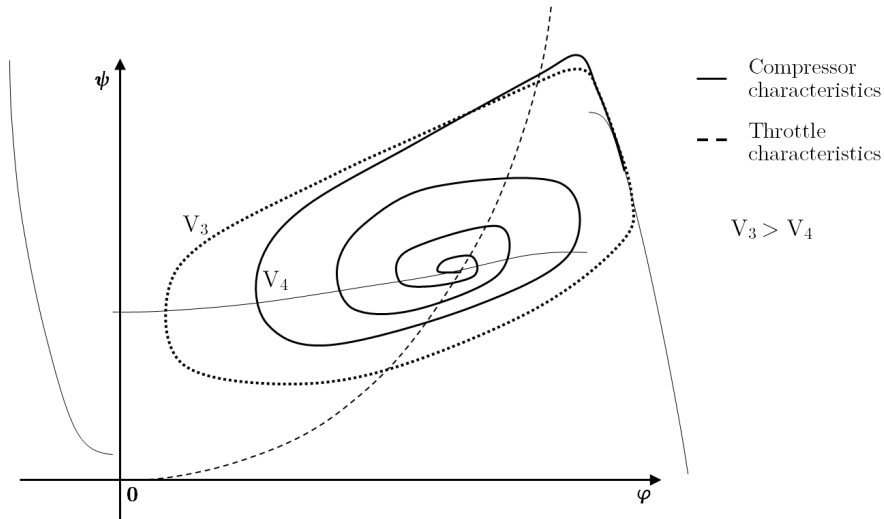


Fig. 3.13: Trajectories of classic surge and mature rotating stall development.

The rotational speed of a compressor can also strongly influence the type of compressor stall which occurs. Day and Freeman [14] tested the 8-stage Rolls-Royce

VIPER compressor stalling it at different rotational speeds and observed different types of stall, figure 3.14. This testing was done on a real engine with a working combustor downstream of the compressor.

At the lowest speed the front stages would experience a multi-cell, part-span rotating stall which is of light intensity and is not dangerous for the compressor or the engine's health. At higher speeds (in the case studied, 65% to 85% speed) a full-span, single-cell rotating stall is experienced. At even higher speeds surge will occur and at the very highest speeds during the flow reversal the combustor will flame out and the engine will shut down.

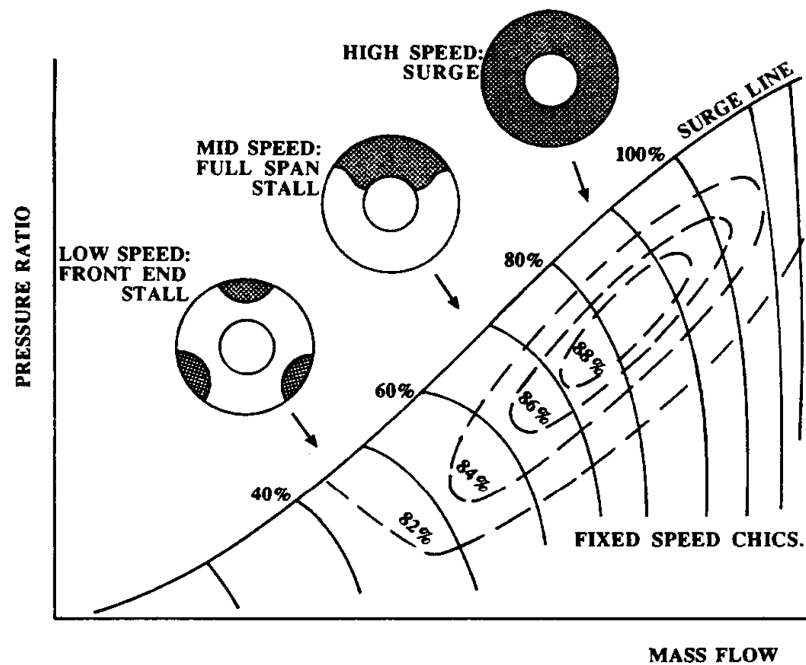


Fig. 3.14: Stall inception behaviour of the Rolls-Royce VIPER compressor at different speeds [14].

In this test an effort was made to ensure that the rotational speed was constant as the engine was artificially throttled beyond the stability line. As can be seen in figure 3.14, at a higher rotational speed corresponds a higher peak pressure ratio when stall occurs. As Day suggested [58], the different types of stall should be linked to both the rotational speed *and* the pre-stall pressure ratio, which represents the amount of energy stored in the pressurised combustor.

Greitzer [33] analysed the system stability on a lumped compressor model and proposed a non-dimensional parameter B to determine whether it would surge or

get locked in rotating stall. The Greitzer parameter is as follows:

$$B = \frac{U}{2a} \sqrt{\frac{V_{Pl}}{A_C L_C}} \quad (3.1)$$

where U is the mean blade velocity, V_{Pl} is the volume of the downstream tank (plenum), A_C is the effective area of the compressor and L_C is the effective length of the ducting. According to Greitzer's theory for every compressor a critical value B_{CRIT} exists such that if the B parameter calculated for the current system is $< B_{CRIT}$ the compressor will exhibit rotating stall, otherwise it will surge. There is also a value of B above which the surge will be a deep surge and below which it will be a classic surge. When the velocity is kept constant this theory follows the results observed experimentally changing the tank volume. Similarly, when the plenum volume is constant and the rotational speed changes, the same trends from the Rolls-Royce VIPER test are obtained.

The B parameter is widely used in compressor stall studies but the critical value of B parameter can not be predicted for a newly designed compressor. In fact as pointed out by Day [58] the parameter contains only the blade velocity, and no other consideration of the compressor performance (characteristic shapes, peak pressure ratio...). Therefore the value of B_{CRIT} found for a compressor can not be transferred to a different one; only experimental testing can determine a new value.

Many other factors are ignored when this parameter is applied to real engines. In experimental testing the surge is often initiated by closing the throttle until the stability limit is reached. In cases where the stall is triggered in some other way, B_{CRIT} can be different. For example, in case of a VSV malschedule or FOD the compressor performance will be modified and the value of B_{CRIT} would change unpredictably. As a result the B parameter is of limited use in predicting the stall behaviour during the design of a new engine.

Linearised model of compressor stall dynamics and stability

The lumped model proposed by Moore and Greitzer (described in section 3.2.1) can be used to investigate the stability and surge dynamics of a compressor rig with

compressor, plenum and throttle. Paduano [5] proposed to study the compressor rig system using an analogy with a mass-spring-damper system, as shown in figure 3.15.

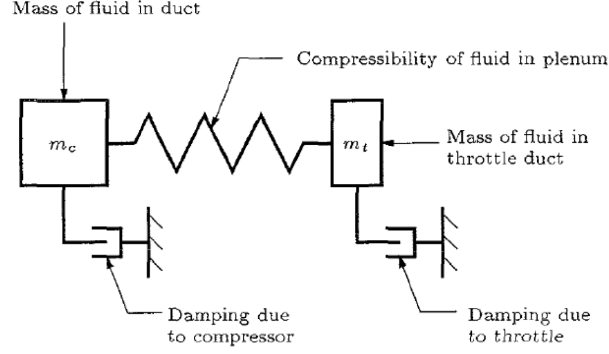


Fig. 3.15: Mass-spring-damper system analogy of the Moore and Greitzer lumped model. Reported by Zoppellari [11].

In this analogy the masses are the fluids in the compressor and throttle duct, the spring is the pressure build up in the plenum and the compressor and throttle provide the damping.

In the Moore-Greitzer model compressor and throttle are modelled only through their characteristics. The equilibrium point of a compressor-plenum-throttle system is found at the crossing of the compressor and throttle characteristics (figure 3.16). The characteristics are linearised around the equilibrium point and we define their slopes as:

$$\mathcal{C}_C = \left. \frac{d\Delta P_C}{d\dot{m}_C} \right|_{EQUILIBRIUM} \quad (3.2)$$

$$\mathcal{C}_T = \left. \frac{d\Delta P_T}{d\dot{m}_T} \right|_{EQUILIBRIUM} \quad (3.3)$$

If we consider the system in figure 3.15 oscillating around the equilibrium point, its motion can be described by the characteristic equation

$$s^2 + \frac{1}{L_C} \left(\frac{1}{4B^2\mathcal{C}_T} - \mathcal{C}_C \right) s + \frac{1}{4B^2L_C} \left(1 - \frac{\mathcal{C}_C}{\mathcal{C}_T} \right) = 0 \quad (3.4)$$

where the displacement modelled is the mass-flow (or ϕ) in the compressor, B is the Greitzer parameter and L_C is the length of the compressor.

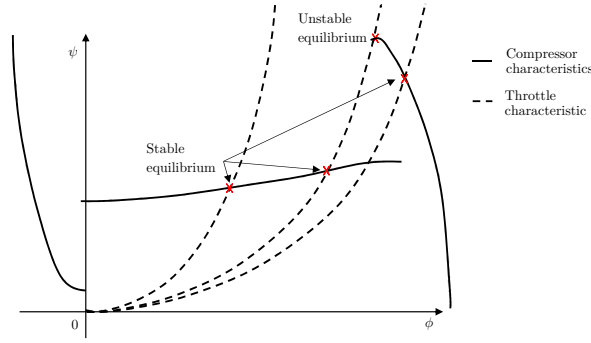


Fig. 3.16: Equilibrium points of a compressor rig at different throttle settings

Using equation 3.4 we can analyse the stability of the possible equilibrium points of this system. The system is stable when the coefficients of elasticity and damping are both positive, therefore:

$$\mathcal{C}_C < \mathcal{C}_T \quad (3.5)$$

$$\mathcal{C}_C < \frac{1}{4B^2\mathcal{C}_T} \quad (3.6)$$

Since the throttle characteristic slope and the B parameter are always positive, a negative sloped compressor characteristic is always stable. As can be seen from figure 3.16 the first condition is always satisfied at every crossing point. The second condition is satisfied for negative $\mathcal{C}_C$ but also for a small range of positive characteristic slope. In a typical compressor this range is very small ($4B^2\mathcal{C}_T$ is a large value) and the compressor will become unstable at $\mathcal{C}_C \sim 0$. Based on this criterion the only parts of the characteristic which are stable are the negative sloped one and the rotating stall characteristic if $\mathcal{C}_C$ is low enough.

Hynes and Greitzer [29] studied compressor stability using a more complex version of this model where the flow is two dimensional and described using a Fourier transformation. While the axi-symmetric characteristic predicted the stability limits exactly as described in equations 3.5 - 3.6, they also found that any harmonic disturbance in the flow would become unstable (and grow) when $\mathcal{C}_C < 0$. These instability analysis seems to support the experimental findings on *modal* stall inception, which is in fact described as the growth of asymmetrical perturbations once the compressor is throttled slowly beyond $\mathcal{C}_C = 0$.

It is important to remember that this linearised stability analysis is valid for compressor which reach the peak at $\mathcal{C} = 0$ *unstalled* (i.e. without any blade passage separating). Modern high-speed compressors generally become unstable because of a local separation and not because of the mismatching of the compressor and throttle/NGV.

Equation 3.4 can also help in understanding the shape of the mass-flow oscillations during surge. If we consider a sharp throttle closure (and relative characteristic change) this will set an equilibrium point with the compressor characteristic in the stalled region. This can be either the axi-symmetric or the rotating stall characteristic, depending on the flow-field in the compressor. In this condition the use of a linearised analysis is not appropriate but, as proposed by Paduano, it can give some qualitative description of the effects of the single parameters on the dynamics. The coefficients in equation 3.4 are the damping and the elasticity of the system and they both contain $B, \mathcal{C}_C, \mathcal{C}_T$. A large B parameter will result in a small damping and a weak elastic response, and therefore large flow oscillations,; a small B will instead cause small oscillations, figure 3.17.

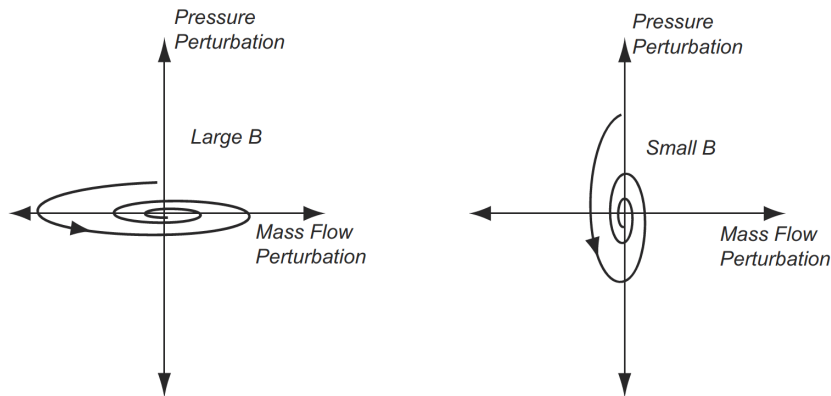


Fig. 3.17: Transient response of surge system to a pressure perturbation [5].

Although this was not reported by Paduano it seems reasonable to extend the analysis and observe that a compressor characteristic with a steep slope will have small damping and a weak elastic response. This observation is particularly meaningful if we consider that typically the $\mathcal{C}_C$ of an axi-symmetric characteristic is always higher than that of a rotating stall characteristic. Therefore, a compressor in rotating stall generally exhibits smaller oscillations and more damping than a compressor

which has not developed any asymmetry. This observation will be useful during the analysis of some of the results from the new tool, in chapter 6.

3.1.5 Compressor stall recoverability

As the compressor completely unstalls during a surge cycle it is normally quite simple to recover from it. It is in fact sufficient to avoid the conditions which caused the crossing of the stability line at the previous cycle. If the initial perturbation was temporary and disappeared, the compressor will recover automatically (e.g. a fuel spike). If a damage occurred and the surge margin was reduced the compressor can be recovered by opening the bleed valves and lowering the operating point on the speed-line. This is in part equivalent to opening of the throttle in a compressor rig (see figure 3.11).

In case of a rotating stall instead the hysteresis of the cycle is such that even opening the bleed valves is often not sufficient to return to unstalled condition. In these extreme cases it is necessary to reduce the rotational speed to recover and then re-accelerate; sometimes it is even necessary to turn off the engine and restart it.

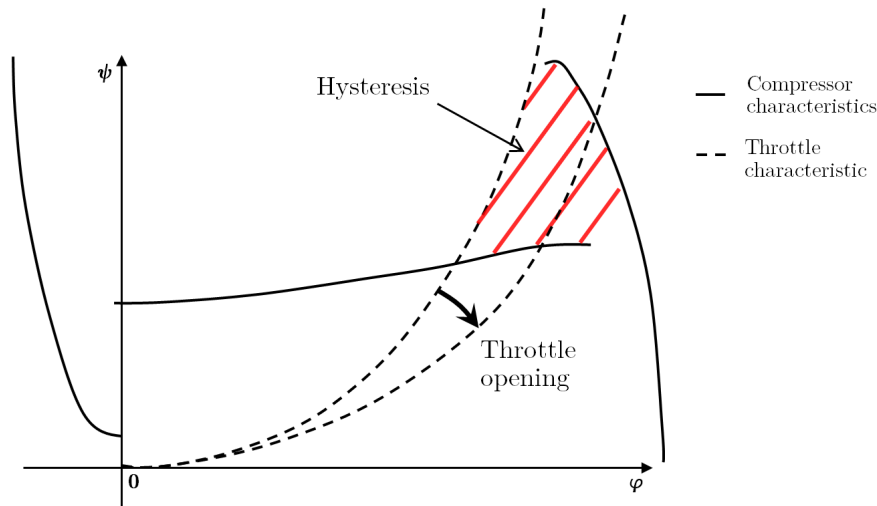


Fig. 3.18: Hysteresis in rotating stall and effect on recoverability.

The analysis of surge dynamics reported in the previous section also suggests that a rotating stall equilibrium, because of the flat characteristic, is very stable

and responds to perturbations with small, damped oscillation. For this reason it is more difficult to destabilise a rotating stall equilibrium point and trigger a recovery.

3.2 Analytical/numerical modelling of compressor stall

In this section a review of numerical methods used to model compressor stall is presented. For the most meaningful cases a description is given of the specific features and results obtained; these are compared together in the summary to analyse the state of the art and identify which methods offer the best capabilities. The different modelling methods are here presented in increasing order of complexity.

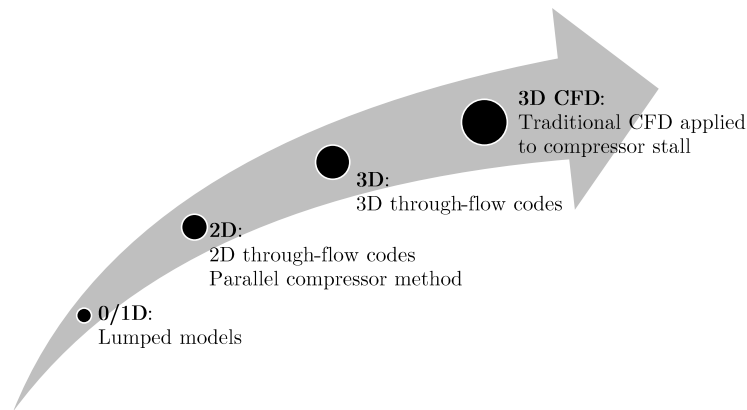


Fig. 3.19: Compressor stall models in order of complexity and computational cost.

A particular type of low-order codes used for modelling of compressor stall are *through-flow codes*. Through-flow codes have been used in the design of turbo machinery since the 1960s and are still used now as fast alternative to 3D CFD, especially in the case of multi-stage compressor [30]. In this type of code the flow-path is solved as a one, two or three-dimensional domain. Unlike normal CFD the blades are not present in the domain geometry but their effect is imposed on the flow. The domain becomes simply the empty annular duct. The flow is usually modelled using Euler equations and every viscosity is introduced as a blade effect.

The effect of the blades is often introduced as a turning and a pressure loss, which requires the estimation of deviation and pressure loss of the blade rows. In

alternative it can be introduced as a pressure and temperature rise, which must also be estimated. Usually this is done by means of empirical correlations or characteristics. Typically, characteristics are used when the compressor performance is modelled as a whole, correlations instead are used to model performance of single blade rows or stages. There are many effects which contribute to the deviation and pressure loss of a single blade row, figure 3.20, and trying to estimate them with a correlation requires strong simplifications. Many sets of correlations have been developed, mostly based on linear cascades experimental testing, but none of them has a universal application.

The effect of blade can be practically implemented in the equations in different ways. The most popular method is the introduction of forces as source terms, to impose the right turning and pressure loss. This is called the *body force method*.

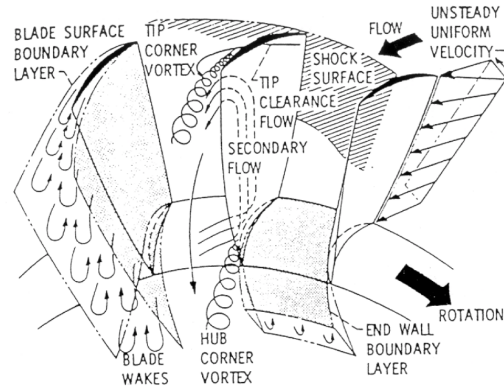


Fig. 3.20: Sources of pressure loss and deviation in a cascade row.

3.2.1 0D/1D models

Through-flow codes

One dimensional (mean-line) through-flow codes are broadly used in the preliminary stage of compressor design, however they are not typically used for post-stall modelling. An exception is the one-dimensional mean line through-flow code developed at P.I. Baranov Central Institute for Aviation Motors. This code has been further developed to model steady reverse flow and rotating stall by Mikhailov et

al. [31]. The tool models the stages performance based on ‘compressor stage generalised functions’; this functions are developed by a statistical analysis of empirically found compressor stage maps. Experimental data were collected by many papers for steady reverse flow, full span mature rotating stall and part span mature rotating stall. The rotating stall is assumed to be part-span or full-span depending on the level of blockage (or the flow coefficient). The code was validated reproducing a low-speed 3-stage rig’s compressor map data, but the matching was not accurate. Despite being able to model even a three-dimensional stall (part-span stall) with a one-dimensional tool this method requires a large knowledge of detailed performance of similar compressors to build up the functions and can only model steady conditions.

Lumped models

Lumped models were the first tools used to model compressor stall and the idea was initially proposed by Greitzer [33].

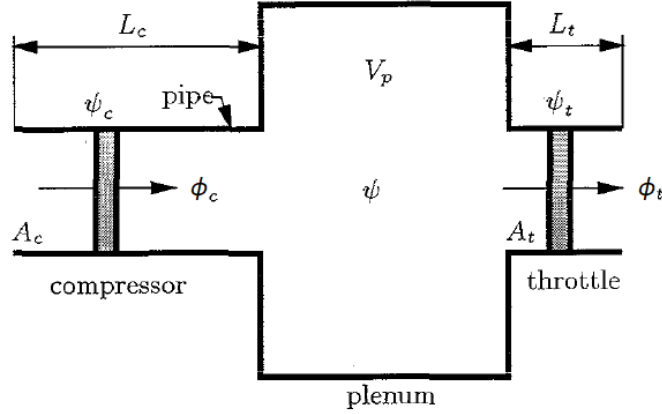


Fig. 3.21: Zero-dimensional lumped model created by Greitzer [33].

In this lumped model the compressor is substituted by a total pressure rise function,

$$\psi_C = f(\phi_C) \quad (3.7)$$

which is taken from the experimentally measured characteristics. In the original version only the rotating stall characteristic was used in the post-stall region. The flow is considered incompressible in the compressor and the throttle. The plenum

downstream is modelled as a volume with uniform pressure and isentropic processes. It exhausts to the atmosphere through a throttle which is defined by a parabolic characteristic

$$\psi_T = f(\phi_T) = \frac{1}{2}K_T\phi_T^2 \quad (3.8)$$

The plenum properties depend on the mass-flow exchanged with the compressor and through the throttle:

$$\frac{d\rho_{Pl}}{dt} = \frac{1}{V_{Pl}}(\dot{m}_C - \dot{m}_T) \quad (3.9)$$

$$\frac{dP_{Pl}}{dt} = \frac{\gamma P_{Pl}}{\rho_{Pl}V_{Pl}}(\dot{m}_C - \dot{m}_T) \quad (3.10)$$

The momentum equation along the axial direction in the compressor and throttle, and the plenum equations are non-dimensionalised using two parameters B and G . The equations become:

$$\frac{d\phi_C}{dt} = B(\psi_C - \Psi) \quad (3.11)$$

$$\frac{d\phi_T}{dt} = B/G(\Psi - \psi_T) \quad (3.12)$$

$$\frac{d\Psi}{dt} = \frac{1}{B}(\phi_C - \phi_T) \quad (3.13)$$

where Ψ is the pressure ratio across the compressor $\Psi = P_{Pl}/P_{ATM}$. B is the Greitzer's parameter

$$B = \frac{U}{2a} \sqrt{\frac{V_P}{A_C L_C}} \quad (3.14)$$

and G is the ratio of compressor and throttle inertia defined as:

$$G = \frac{L_T A_C}{L_C A_T}. \quad (3.15)$$

Normally the length of the throttle L_T is negligible and G becomes very small. The throttle then adapts almost instantaneously to the changes, $\psi_T = \Psi$, and the throttle equation is cancelled. The values of pressure rise obtained from the experimental characteristic are valid for steady state. In order to extend them to transient simulation the pressure rise in the equations is updated using a first order

time lag law,

$$\frac{d\psi_C}{dt} = \frac{1}{\tau}(\psi_{C,ss} - \psi_C) \quad (3.16)$$

where the time constant τ is based on the rotation period

$$\tau = \frac{2\pi r}{U} \quad (3.17)$$

The original model by Greitzer was used to simulate classic surge for a low-speed compressor.

A model aimed only at modelling steady, mature rotating stall was later developed by Moore and was then integrated with the model from Greitzer to create a code which could deal with both the instabilities and their developments [34]. In this code the flow coefficient is a function of time, axial and circumferential location $\phi(t, x, \theta)$. The flow is considered uniform and irrotational at the inlet and thus in the compressor system the flow coefficient can be described as a sum of a circumferentially averaged value $\bar{\phi}$ and harmonic perturbations

$$\phi = \sum_{n \neq 0} \tilde{\phi}_n(t) e^{in\theta} e^{|n|x} + \bar{\phi}. \quad (3.18)$$

The compressor pressure rise instead is evaluated at the compressor's front face and is also specific of the circumferential position, $\psi_C(t, \theta)$. The compressor characteristic used in this model is only the axi-symmetric one: a single line which comprehends unstalled, stalled and reversed regimes. The new model was successfully used to predict surge and rotating stall with good agreement with the experiments [34, 35]. Longley [23] in his review of lumped codes reported how these models can also predict the cell propagation speed based on the blade rows' inertia.

Despite the results obtained, these 'lumped' models could never distinguish between part-span or full-span stall, and could not have any indication whether the stall is incepted at the front or the rear of the compressor. Also they could not distinguish if a rotating stall propagated axially along all the stages or was confined in a portion of the compressor. Furthermore the lumped model can not be used to predict the performance of the compressor during stall, but on the contrary it

requires as an input the axi-symmetric and reverse flow characteristics.

3.2.2 2D models

Meridional through-flow codes and SLC codes

2D meridional through-flow codes are routinely used for performance prediction. While they can model flow reversal they can not simulate rotating stall as the flow is assumed symmetric. For this reason there is no meaningful example of this type of code used for post-stall modelling. For the present project only their set of empirical correlations for forward flow are of interest. The through-flow codes from Al-daini [36] and more recently Petrovic [37] which contain various sets of empirical correlations were reviewed although in the end these were not used.

A method which has many features in common with meridional through-flow codes is the Streamline Curvature Method (SLC). The SLC models the flow in the compressor divided in streamlines which do not exchange mass-flow but change their area along the axial direction. Calculations are performed in the meridional plane and are generally used to obtain steady state performance. The code SOCRATES is a SLC code developed in Cranfield for fast prediction of compressor performance in a large range of operating conditions [38]. Portions of this code have been used in the present tool, in particular the set of empirical correlations for unstalled flow and the algorithms which perform annulus and blade geometry creation.

2D circumferential through-flow code

Hendricks et al [39] proposed a 2D circumferential through-flow code where the blades were represented by actuator disks whilst every inter-blade space and inlet/outlet duct was modelled with 2D Euler Equations. Bleeding was introduced in the actuator disks. A two-spool high-speed compressor was modelled as validation. The inception of inception was simulated and matched with experimental results, however the model was not able to simulate the post-stall behaviour and no mature rotating stall or surge was modelled.

Lindau and Owen [40] developed a two-dimensional tool where the body force

method was replaced by using two different sets of equations for blade rows and blade-free regions. In the blade-free region the two-dimensional Euler equations were used, inside of the bladed regions the equations were transformed into one-dimensional by restricting the circumferential velocity based on the local passage direction. The solver used was a Roe solver as it was thought to be suited for unsteady flow modelling. Blade performance was estimated using a table of deviation and pressure losses taken from literature. However no experimental validation was presented in the paper and the results appear to be only preliminary.

In her PhD project, which spanned from 2010 to 2012, Zoppellari created a quasi-2D (using parallel compressors) and a fully 2D solver to model rotating stall and surge [11]. A parallel compressor method is when a 2D discretization is achieved by putting together many 1D domain and allowing some flux exchanges between them. The code successfully reproduced the reverse characteristic and part of the rotating stall characteristic of the C106 compressor (see section 3.3). Surge was also modelled and the correct trend was observed although no direct validation with the reported trajectories or frequencies was done.

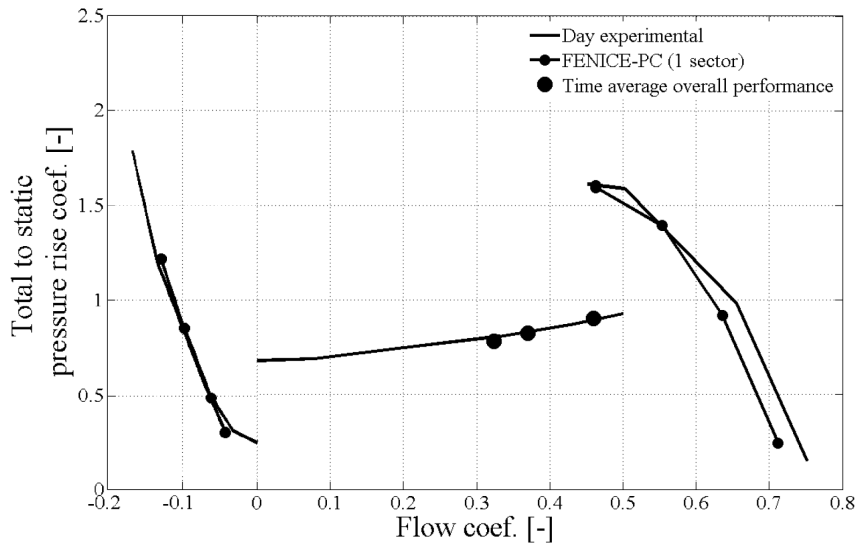


Fig. 3.22: C106 complete compressor map with results from FENICE [11].

The present work is in fact the continuation of that effort with the introduction of the span dimension. Some of the methodology employed in FENICE has been used in the new code, this includes:

- plenum and throttle model,
- correlations for stall detection,
- treatment of the fluid in the relative FOR (rotor blades).

These will be described later in the methodology, chapter 4.

The method employed to model the performance of separated blade passages in FENICE is a lumped version of Longley's method [41], described in section 3.2.3. During reverse flow a total pressure loss was calculated based on the blockage factor b , obtained by integrating the entropy generated by the full mixing of the wake. The total pressure loss coefficient is then a function of the blade stagger angle γ and inlet angle α

$$\omega = \left(\frac{|\sin(\alpha - \gamma)|}{\cos(\alpha)} \right)^2 (\cos(\alpha))^2 \quad (3.19)$$

The deviation was instead set to 0, ignoring the effect of wake mixing on flow direction. During the transient conditions when the flow was neither fully reversed nor fully forward, no correlation was applied. This method of modelling reverse flow performance was initially used in the current project as the author took up the project from where Zoppellari finished. However, when applied to the first test case it was found to have very poor results and a new methodology was developed, this will be described in section 4.7. The finite difference MacCormack scheme from FENICE was also used in the first stages of development of the new code, but it was deemed not robust enough during the transients. It was later substituted by the Godunov scheme.

In 1996 at Cranfield university PhD student Alexander Wilson [42] created two models to simulate the stall behaviour of the Rolls-Royce VIPERTM engine compressor, tested by Day and Freeman [14]. The first model is a quasi-2D code with the compressor modelled as a series of parallel compressors. The flow is considered incompressible and inside of the compressor only the axial direction is solved, the tangential velocity in the compressor is assumed $u_\theta = 0.5 U$. The blade rows are modelled as a *semi-actuator disc*. In this model the whole compressor performance is interpolated from a single characteristic but the body forces (only axial)

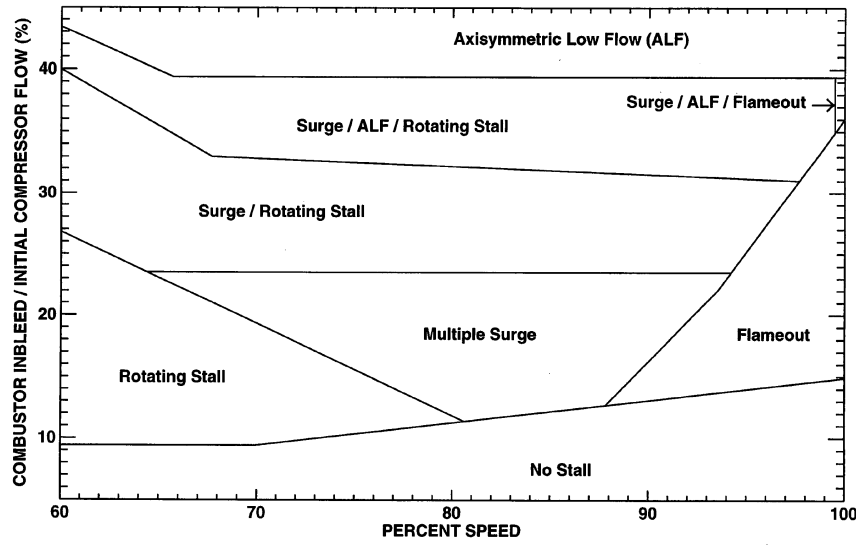


Fig. 3.23: Simulated post-stall behaviours of the Rolls-Royce VIPER compressor [42].

are distributed across the compressor domain to take into account the inertia of the flow inside. Only the forward, unstalled characteristic is experimentally known. The axi-symmetric stalled characteristic is interpolated assuming that at $\dot{m} = 0$ the pressure rise would have been 15% of the peak value. The reverse characteristic is obtained by scaling data from literature, the rotating stall characteristic is not used. The compressor domain is solved numerically while the inlet duct is solved analytically. To simulate the experimental condition also the combustion chamber is modelled using two volumes (cold and hot) and a simple combustion efficiency. Like in the experimental rig simulated, air can be bled inside of the combustor prompting the compressor to stall by reducing the available turbine capacity. A random disturbance is introduced to allow the code to develop asymmetric structures.

The quasi-2D code was successfully used to model most of the behaviours observed experimentally, although a limitation was the 2D semi-actuator disk which did not allow to model rotating stall confined to front stages only, or part span cells. By varying the rotational speed and the flow pumped into the combustor the cases simulated were surge, rotating stall, flame-out and the case where the flow pumped in the combustor exits steadily from both the turbine and the compressor (called "ALF", axi-symmetric low flow). The results are shown in fig. 3.23. The second model created by Wilson is a 3D model and is described in the next section.

3.2.3 3D models

A 3D body force method was developed at the Massachusetts Institute of Technology starting with the work of Gong [43] and improved in the subsequent years. The initial tool was created to simulate rotating stall and employed a body force method based on a database of correlations produced using CFD. In this methodology the compressor geometry is simulated using RANS CFD in a large range of operating condition. A database of the blade response to the different conditions, in terms of pressure rise and turning, is created and stored. This is then used to create a body force method which responds dynamically to the changing conditions. The method developed by Gong is valid only for incompressible flow and strictly axial machines. The validation was done on a 4-stage GE compressor and a single stage experimental rotor. It focused on modelling stall inception triggered by localised disturbances (spoiled inlet sectors). The main features of the flow breakdown and cell growth were observed in general accordance with experimental observation, although reverse flow was not modelled and the simulation could not be taken further. Only the steady state results were directly validated but only on a single compressor.

The tool was then further developed by other researchers at the MIT which improved the method making it more general. The works of Kewada, Reichstein, Patel, Walker and Kerner are cited and briefly described by Brand [44], which himself improved the body force method adapting it to compressible flow and non strictly axial streamlines. Brand verified the new solver modelling a fan rotor and matching it with the RANS simulations used to create the database. The unstalled results were successfully matched with the CFD, although the values of entropy creation and turning were taken from the CFD itself. Operation at low mass-flow (i.e. post-stall) was simulated but no validation was possible as the RANS simulation diverged when the flow separated.

A 3D code was created by Wilson to model stall inception in high-speed compressors [42]. This is a through-flow code using Miller's correlation to determine pressure loss and deviation. The solver used is a finite volume method in the meridional plane while a finite difference method was used to calculate the effects in the

circumferential direction. The correlations for reverse flow are not introduced and as such the code can not model post-stall behaviour but only stall inception. A quite ambitious feature of the code is that it solves RANS (Reynolds averaged Navier-Stokes) instead of Euler equations. This is done following the work by Gallimore [45], which first introduced the shear stress in a 2D meridional through-flow code. However, unlike the previous work, not only the radial variations in velocity are considered in calculating the shear stress but every term in the meridional plane. The use of no-slip condition at the end-wall produces the boundary layer; the grid is not uniform but finer close to the walls for better resolution. The body force method is relatively simple (compared to similar codes) but includes the effects of tip clearances by setting the forces at zero in the end wall region. The method was tested to prove its unsteady and near wall modelling capability, however it was validated for stall inception against only one test case.

Escuret and Garnier developed a 2D and a 3D code for surge and rotating stall modelling at SNECMA [46]. The 2D code is an axi-symmetric code where the blades are solved using a 2D through-flow method developed at ONERA. The 3D version instead uses fully 3D equations only in the blade-free volumes and multiple 2D (meridional) through-flow solvers for the bladed regions. The bladed regions are solved in a rotating frame of reference and a blockage is introduced to take into account the fraction of the annular area occupied by the blades. The solver used is a Jameson-style multi-step finite volume method coupled with a residual smoothing technique to increase the CFL up to 10. An atypical type of plenum model is used, a 1D continuation of the computational domain shaped into a large volume, figure 3.24.

A single-stage case was reported. The domain of about 60000 points (113 X 12 X 47) would take 3 hours to simulate 0.1 s of physical time on a single core. The code was used to predict the stability of the compressor when perturbed by a sinusoidal or axi-symmetrical perturbations. Only the instability limit was validated while the details of the flow-field were not. Rotating stall was obtained for different mass-flows but the code was not adapted to deal with reverse flow, and only classic surge of small oscillations was modelled. The results reported appear to be only preliminary

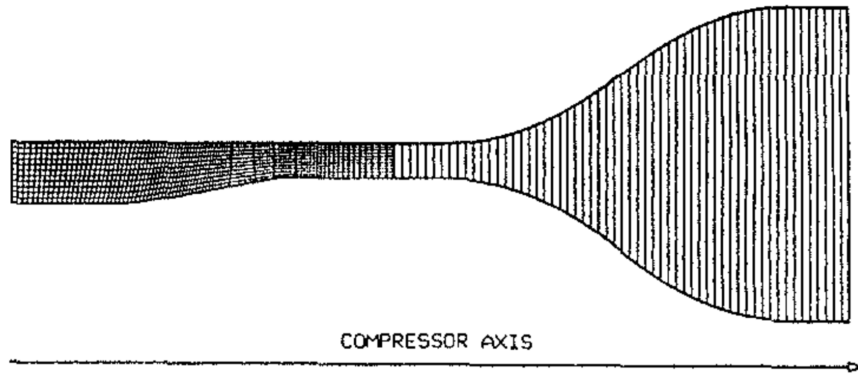


Fig. 3.24: Example of compressor and plenum domain as modelled by Escuret and Garnier [46].

and the validation is based only on the absence of clearly mistaken patterns.

Through-flow codes with separated flow modelling

At Cambridge University Longley created a 2D model similar to the one proposed by Escuret which was used to model the stall inception and how inlet distortion would affect it[47]. This was later expanded into a 3D FVM code capable of simulating the post-stall performance [41]. The main contribution in his paper is a new method to model directly in the solver the highly separated flow which occurs when the blade passages stall. He introduced a variable b to quantify the flow non-uniformity in the length-scale of the blade channel, which is smaller than the domain grid itself. The necessity of the term b arises from the fact that a grid element corresponds to one or more blade passages; when the flow in the passage is separated it becomes impossible to describe mass-flow and momentum with the primitive variables. In a 1D case the primitive variables are $V = [\rho, u, T]$, if the flow is uniform the mass-flow $\dot{m}$ and momentum Q can be described as

$$\dot{m} = \rho u, \quad Q = \rho u^2. \quad (3.20)$$

However, when the flow is non uniform in a length-scale smaller than the element itself, it becomes impossible to simultaneously describe mass-flow and momentum with these terms alone. An example of this situation is shown in figure 3.25.

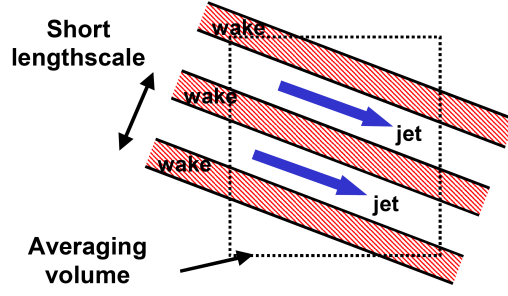


Fig. 3.25: Flow separated at a lengthscale smaller than the grid element [41].

If we consider the density to be constant in the element an equivalent velocity $\bar{u}$ can be defined, which has the same mass-flow as the separated flow:

$$\dot{m} = \int_V \rho u \, dV \Big/ \int_V dV = \rho \bar{u}. \quad (3.21)$$

The momentum however will always be higher then the value that a uniform flow would have

$$Q = \int_V \rho u^2 \, dV \Big/ \int_V dV \geq \rho \bar{u}^2, \quad (3.22)$$

the ratio of the LHS over the RHS of this disequation is always higher than 1 and it's the parameter b . The real mass-flow *and* momentum can then be described if b is added to the primitive variables

$$Q = \rho b \bar{u}^2. \quad (3.23)$$

With the new variable it is necessary to modify the Euler equations and a sixth equation is introduced to model the transport, creation and dissipation of the non-uniformity. The conservative variable used is the *excess energy* which is the difference between the kinetic energy of the real flow and that of an equivalent uniform flow with the same mass-flow. The three-dimensional form of the equation is:

$$\frac{\partial}{\partial t} \left(\frac{1}{2} \rho (b-1) \bar{u}^2 \right) + \frac{1}{2} (b-1)^2 \bar{u}^2 \nabla \cdot (\rho \mathbf{u}) + \left(\frac{b-1}{b} \right) \rho \bar{u} \nabla \cdot \left(\frac{1}{2} b^2 \bar{u}^2 \right) = -\mathbf{u} \cdot \mathbf{F} - \rho T \frac{DS}{Dt} \quad (3.24)$$

The RHS terms represent the creation and dissipation of the non-uniformity in the flow, the first term generates separation every time the blade forces are not perpendicular to the flow. The last term is an entropy creation term which controls the dissipation of the non-uniformity, it is calculated as:

$$\rho T \frac{DS}{Dt} = \left(\frac{b|\bar{u}|}{1/2s} \right) \frac{1}{2} \rho (b-1) \bar{u}^2. \quad (3.25)$$

The ratio of dissipation (in time) is set as the time for the jet flow ($b|\bar{u}|$) to travel the distance of half a span. Despite being a somewhat arbitrary choice, it is justified by the fact that whichever is the length of mixing the integral of the energy dissipated is always the same.

Through the creation, transport and mixing of the non-uniformity it was possible to naturally model pressure loss and deviation during reverse flow and transients. Rotating stall and surge was then modelled without any need for empirical correlations except for the forward unstalled flow ones. The reversed and rotating stall characteristics of the 3-stage compressor described by Gamache and the 4-stage compressor tested by Day (see section 3.3) were reproduced and matched with experimental results although not for the whole range of flow coefficients. Surge was modelled although not validated against any experimental finding. The domain used for the 4-stage compressor had 131 axial $\times$ 6 radial $\times$ 32 circumferential elements and required 30 hours of computational time for 2 s of physical time on a single core.

3.2.4 3D CFD

Classic CFD codes are routinely used to model the performance of turbomachinery and in the years have earned a great trust in their reliability and accuracy. However when modelling multi-stage compressor systems a great computational cost is required. In fact it is normal practice to simulate only a reduced amount of blade passages instead of the whole annulus to save time and resources. As part of the shaft failure and over-speed studies in Cranfield, Lucas Pawsey [10] modelled the reverse flow in a multi-stage modern compressor. The domain was circumferentially only 1/5 of the annulus and consisted of 32 millions elements, the program used

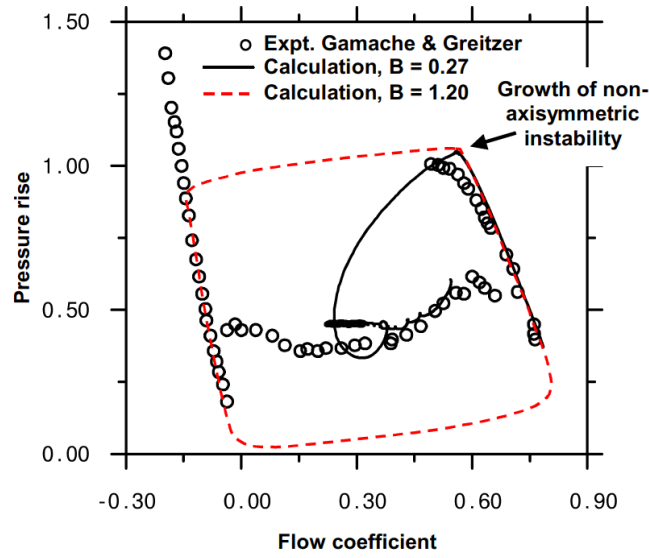


Fig. 3.26: Surge and rotating stall of Gamache's 3-stage compressor modelled by Longley [41].

was ANSYS CFXTM. To properly model the unsteadiness in the separated flow a transient simulation was used. The computational power required was very large and the simulation run on 64 dedicated cores for 2 months before reaching a steady state.

3D CFD has been also used to model rotating stall although enormous computational resources are required. One of the latest and most state of art attempts is the work done at London Imperial college and reported by Dodds and Vahdati [19, 48]. A modern 8-stage intermediate-pressure compressor was tested experimentally at a range from 50% to 80% design speed and throttled to stall. The unsteadiness obtained was a front-end multi-lobed rotating stall. The compressor was then modelled using the code AU3D in order to simulate the same stall. As the stall is present in the front part only, the first three stages were modelled in 3D with full annulus while the rear ones were modelled as a one-dimensional domain. The 3D grid had $\sim 250,000$ elements per blade passage and when processed on a cluster with 160 cores it took 2 days per rotor revolution. The rotating stall was validated against experimental results for the full range of velocities, providing accurate and detailed description of the stall cell structure. In figure 3.27 the development of a multi-cell part-span stall, caused by VSV movement, is visualised at different moments in

time.

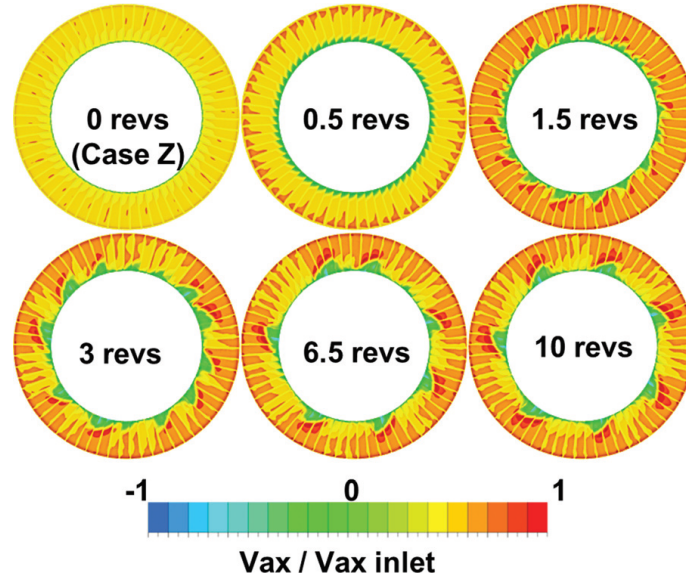


Fig. 3.27: Part-span stall cells development, following a stator vane restaggering [19].

3.2.5 Compressor stall modelling methods summary

All the models reviewed in this chapter are compared in table 3.1; only the models capable of simulating rotating stall are considered. The features compared are:

- the presence of span-wise resolution,
- the capability to model reverse flow,
- if the compressor performance is lumped or modelled stage by stage/by blade rows,
- the method used to model the compressor/stage/blade row performance (labelled ‘*Performance*’)
- the computational time; some older methods which were originally reported to take hours have been put as ‘low’ considering the advance in computational power in the last 20 years
- the capacity to model rotating stall or surge, a distinction is done if the stall was only obtained (o) or also validated experimentally (v).

Model	Span resolution	Reverse flow	Stage-by-stage	Performance	Comput. time	Rotating stall	Surge
Lumped (Paduano)	x	v	x	chic	low	v	v
Hendricks	x	x	v	correl.	low	o	o
Lindau et Owen	x	x	v	correl.	low	o	o
Escuret et Garnier	v	x	v	correl.	hours	x	o
Zoppellari	x	v	v	correl.	hours	v	o
Wilson (2D)	x	v	x	chic	low	o	o
Gong/Brand	v	x	v	CFD	hours	x	x
Longley	v	v	v	correl.	hours	v	o
3D CFD	v	v	v	N/A	months	v	v

Tab. 3.1: Comparison of rotating stall and surge models in literature.

Despite the unmatched resolution and the validated methodology the 3D CFD requires excessive computational resources; the objective of this PhD is indeed to develop an alternative to model three-dimensional compressor stall with lower computational cost. The lumped models have been validated and employ minimal computational resources, however they require the knowledge of the stalled characteristics and as such can not be predictive.

From the comparison of through-flow codes it appears clear that only the three-dimensional code can properly identify the different type of stall (part-span or full-span). However, most of the 3D codes were not validated or even developed to the point of being capable of modelling all types of compressor stall. The tool which is capable of modelling compressor stall with the maximum detail without requiring the knowledge of the characteristic is the one created by Longley. This model also presents the most physically true and generally applicable method to model the performance of a separated blade row. However the addition of a new, singular equation complicates the Euler solver and increases the computational time. Besides the new complex set of equations is incompatible with some particular type of Euler

solver, unless the solver itself is extensively modified. One of these incompatible solvers is the Godunov scheme, which has been chosen to develop the new tool, as is described in section 4.2.2.

Following this review this author decided to:

- use a three-dimensional body force method to model the blade rows adapting one or more of the numerous formulations,
- model the performance of each blade row using empirical correlations to model forward flow and determine when the blade passage stalls
- the performance of blade rows with reverse flow will be based on the separated flow model. However it will not be introduced with an additional equation, but with a simpler correlation.

These choices have shaped the new methodology developed, described in details in chapter 4.

3.2.6 *Full engine stall modelling*

As described in the section 1.2 in order to certify an aero-engine it is necessary to predict the maximum speed reached during shaft failure or to determine whether the shaft failure will lead to rotor burst. The maximum speed reached is not necessarily the terminal speed, depending on the rapidity of the safety systems in shutting down the engine. Specific tools have been created with the objective of modelling the engine during a shaft failure and calculate the maximum rotor speed during the transient. The objective of these tools are:

- Determine the outcome of a shaft failure event, distinguishing between rotor burst or safe run-down.
- Predict the maximum speed during the event, and the speed profile during the transient.
- Determine the effect of changes in engine design on the maximum speed during shaft failure and investigate which design choices can be used to reduce speed.

Haake et al. [49] proposed in 2011 a tool to simulate the engine during shaft failure. This tool uses a set of sub-models to simulate the performance of the components critical to the shaft failure transient.

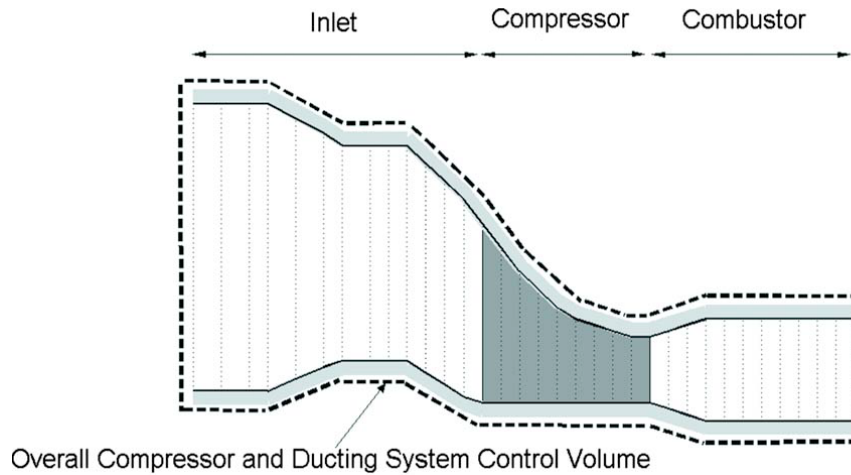


Fig. 3.28: Compressor/combustor domain of the shaft failure modelling tool [49].

Compressor, combustor and inlet ducting are modelled as a single 1D domain, solved using an Euler solver. The domain extends from the core inlet to the first turbine NGV inlet, figure 3.28. The blade rows presence is introduced by an axial force and a work exchange term in a similar fashion to the body force method. These terms are estimated from the pressure and temperature rise extracted from maps. The pressure and temperature rise produced by the compressor are evenly distributed in the domain elements which are part of it. The instantaneous operating point on the map is interpolated based on the mass-flow at the compressor inlet. The inertia in the change of compressor performance is introduced by lagging the pressure and temperature rises. Bleed flows are introduced and modelled as constant source terms. A set of typical stage compressor characteristics are used, each divided into reverse flow, unstalled and rotating stall characteristic. The axi-symmetric characteristic is not used and from the paper it would appear that the rotating stall characteristic is used for both rotating stall and surge.

The heat release in the combustor is calculated using a 0D model and introduced as a source term. The profile in time of the fuel injected during the event must be known *a priori*. The heat release is halted if the conditions in the combustor move

outside the flammability limits and resumed if they return within the limits.

A zero-dimensional model is used to predict the performance (capacity) of the turbines during over-speed, the turbine characteristics are produced using a performance program. A *friction torque* model is used to predict the friction caused by the interaction of the stator and rotor structures. The friction is function of the speed, the turbine structures' shape and the axial displacement of the rotor blade row. The melting of the structures is taken into account and the shape used as input is changed in time as it deforms. By considering the torque produced by the flow on the turbine and the one caused by the friction, the model can predict the acceleration of the turbine rotor and determine the terminal speed.

Individual components have been validate separately and the tool as a whole was validated in steady state on a 10-stage, high-speed HPC. An IPT shaft failure was modelled on a two-shaft engine and successfully matched to experimental data.

The program WEST (Whole Engine Simulation Tool) is the code developed in Cranfield University by the shaft failure and over-speed team [10], started in 2007. This tool uses a similar set of sub-models as used by Haake et al., but has been further developed to consider other aspects critical to the event. The sub-models and their domains are presented in figure 3.29.

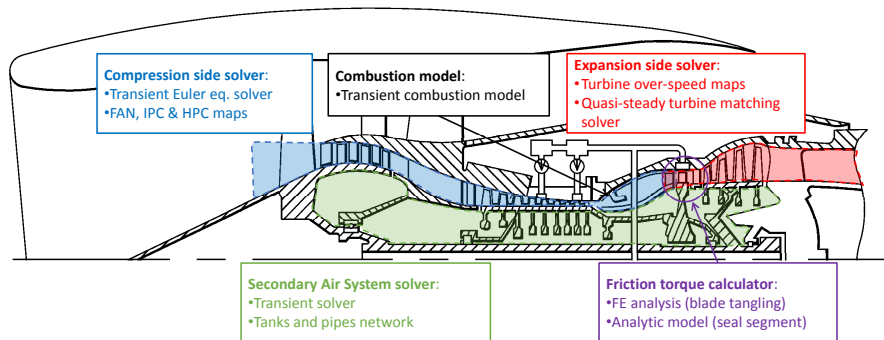


Fig. 3.29: Components of the code WEST [10]. Background image from Plemmons et al. [50].

For brevity only the improvement over the model by Haake will be listed here.

- The compressor/combustor domain extends up to engine inlet, including the lower part of the fan which is modelled with specific maps. The complete maps are required for the compressor, unstalled flow, rotating stall and reverse

flow characteristics. The axi-symmetric stalled characteristic is obtained from the others with the fitting technique described in section 3.1.3. Since the compressor must be able to accelerate or decelerate during transient, maps for different speeds are necessary.

- The SAS (Secondary Air System) is modelled as set of plenums and 1D ducts. The pressure in the SAS determines the bleed flows and is used to calculate the turbine axial displacement. This algorithm dynamically modifies the inputs to the turbine model, the torque model and the compressor.
- The combustor features an improved fuel control schedule which simulates the decisions of the FADEC in real time, in particular the over-speed protection system and the surge protection system.
- The turbines are simulated with a 0D model which considers the HP,IP and LP turbines separately; the turbine maps have been improved to model the over-speed conditions. The turbine performance is now a function of the axial displacement of the blades, incorporating the effects of tip clearance change. Furthermore the different turbines maps are not independent from each other but when a turbine over-speeds it affects the performances of the one downstream.

WEST has been validated at steady state on the full flight envelope on a modern turbofan engine with two compressors. A classic surge caused by a fuel spike was modelled on a single shaft engine and validated with experimental measurements.

While the main objective of WEST is shaft failure modelling, it can also be used to investigate compressor stall as a *whole engine* event. Even though the compressor model is only 1D, the whole engine simulation has some advantage on the post-stall models previously described. In fact it allows to model fuel spikes or slam accelerations as triggers and to assess the interactions of the fan, combustor, SAS and dynamic fuel control on the post-stall transient.

The WEST code employs the Roe scheme as Euler solver on the compressor/combustor domain. It is a first-order Total-Variation-Diminishing finite volume

scheme selected for its robustness and physically true modelling of pressure waves. The Roe scheme was originally developed in Cranfield in 1981 [51]. The Roe solver in WEST has been co-implemented by this author and modified to take into account the source terms and the semi-perfect gas model following the description by Toro [52] and the works done by Mohanraj et al [54] and Di Mare [53]. This scheme does not naturally satisfy the entropy condition (see section 4.2.2), therefore it was adapted to transonic conditions using the algorithm proposed by Harten and Hyman [55]. The code created manages to retain stability and accuracy while being applied to a very coarse mesh with strong source terms and non-smooth solution [10]. The inclusion of all terms and the effect of area variation allows to avoid any unwanted non-conservative effect caused by the interaction of solver and source terms. The solver created can not be applied to 3D while retaining these capabilities and it could not be employed in the present project. However the knowledge developed by the author has been useful to better understand the similar effects occurring in the Godunov scheme employed in this project.

3.3 Experimental investigation of compressor stall

The purpose of this section is to describe the general methods used to experimentally test compressor stall by means of some meaningful examples. The two rigs described here are used as test cases for the validation of the new tool, described in chapter 6.

3.3.1 *Eastland and Gamache's test on the 3-stage compressor*

At MIT Eastland [56] and Gamache [57] tested experimentally a 3-stage low-speed compressor rig for post-stall behaviour. Eastland focused on mature rotating stall while Gamache studied steady reverse flow and mature rotating stall at almost zero mass-flows and negative flows. The compressor has a constant annulus, a hub to tip ratio of 0.88 and blades with aspect ratio close to 1.

The compressor was run electrically, which ensured constant rotational speed, and in both tests a variable area throttle was used to investigate different operating conditions. In order to study a wide range of conditions the throttle exhausted into

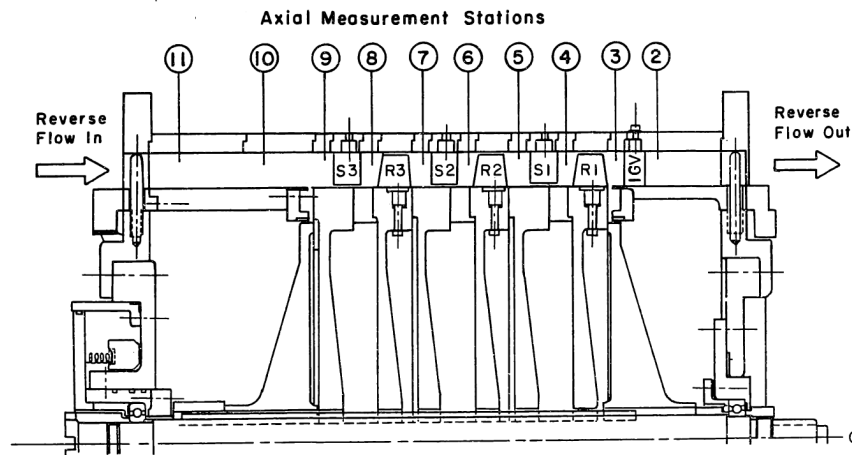


Fig. 3.30: The 3-stage low-speed compressor tested at MIT [57].

a plenum which is connected to the atmosphere through a centrifugal fan. This allowed to obtain back pressures down to sub-atmospheric values in the plenum volume and also to avoid recirculation.

As Gamache's focus was on reverse flow his test was done with the compressor removed and placed in reverse. The sub-atmospheric pressure in the plenum was used to force the flow through the compressor even when running in reverse. The whole test rig is shown in figure 3.31. Gamache run the same compressor geometry of Eastland completing the compressor map with the reverse flow characteristic. A second geometry with a higher reaction was also tested, obtained by re-staggering the blades. In order to obtain performances at zero mass-flow the throttle was modified so that it could be completely closed.

The mass-flow through the compressor was measured by means of an orifice plate: by measuring the pressure drop across it the mass-flow was deducted. Porous plates were used to straighten the flow in the duct before the orifice plate, to ensure no swirl is present and the measurement is accurate. In both the experiments done the static pressure was probed by means of tap sensors on the internal or external wall between every stage. Rakes of total pressure probes were used at inlet and outlet, thermocouples were used in between stages to measure total temperature. The measurement of the axial velocity in unstalled operation, obtained with the

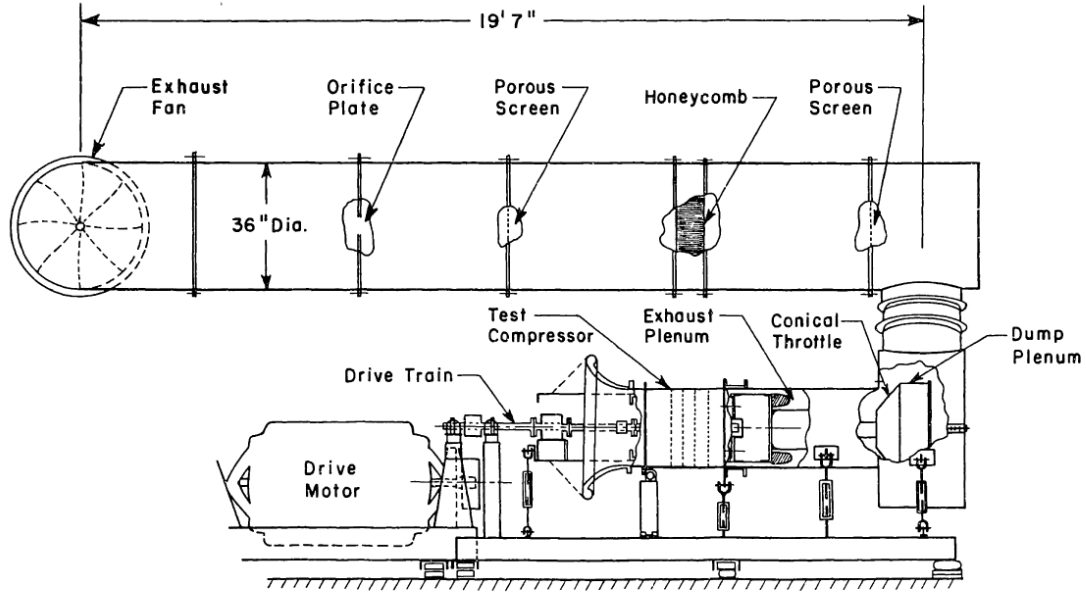


Fig. 3.31: A schematic of Gamache's test rig [57].

pressure probes, was used to calibrate the orifice plate.

Uncalibrated hot wires were used to study the rotating stall. Eastland put one in front the 2nd rotor and one after the last stator while Gamache put only one at inlet of the IGV. These hot wires are monitored on a storage oscilloscope and left a trace which became erratic whenever the wire was inside of the stalled flow, figure 3.32.

It is easy to measure from this trace the period between passages of the cell, T_{RSC} , and the period in which the hotwire was inside of the stalled flow, t_{RSC} . From this data it is possible to calculate both the translational speed of the cell

$$U_{RSC} = \frac{2\pi\bar{r}}{T_{RSC}}, \quad (3.26)$$

the area of the cell

$$A_{RSC} = \frac{t_{RSC}}{T_{RSC}} A_C \quad (3.27)$$

where A_C is the area of the annulus at the axial position where the measurement is taken. Whilst the cell speed is quite accurately calculated, the size of the cell is more ambiguous. It depends in fact on the axial location of the hot wire and on the voltage threshold used to define the period t_{RSC} . As a result the measurements

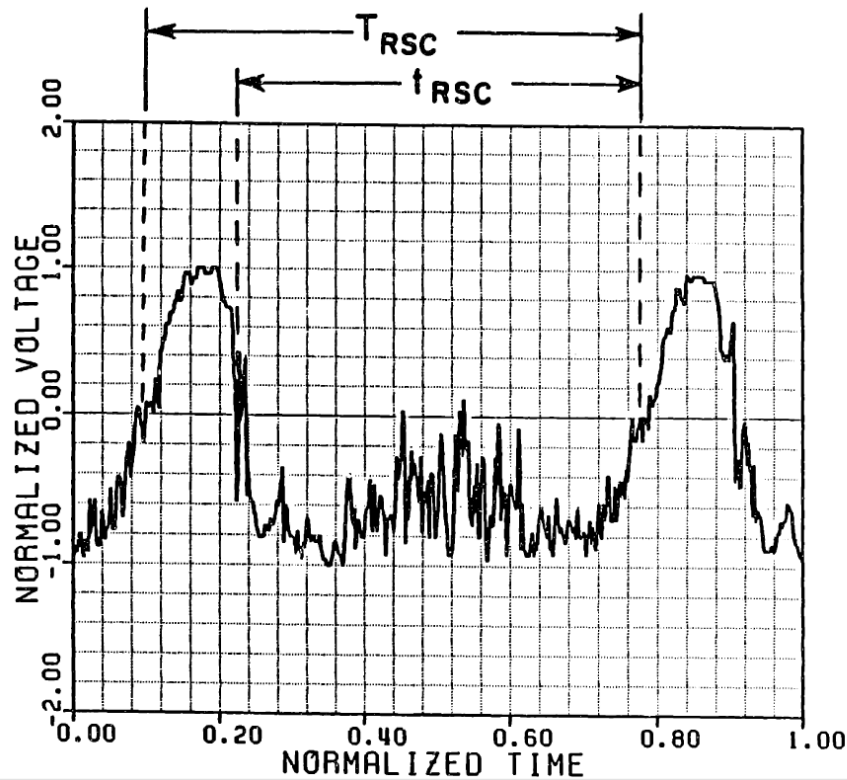


Fig. 3.32: Hot wire trace during rotating stall [57].

of Eastland and Gamache for the same configuration do not match although they follow a similar trend.

3.3.2 Day's test on the 4-stage compressor

At Cambridge a low-speed 4-stage compressor was tested by Day in 1994, [58]. This compressor consists of 4 repeating stages preceded by a high solidity IGV.

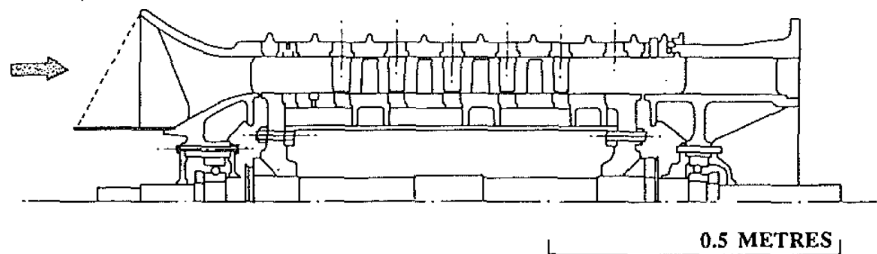


Fig. 3.33: The 4-stage compressor tested by Day [15].

In this set-up the compressor is followed by a plenum volume and a throttle discharging directly into the atmosphere. The plenum and the ducting had variable

volume and length and Day reported results for B parameters ranging from 0.3 to 1.5 (and plenum volumes up to 33 m^3). With the original set-up the compressor would exhibit only deep surge or rotating stall. It was further modified by introducing plates along the exit duct. This reduced the hysteresis of the cycle and permitted to observe classic surge. A set of 6 traverse hot wires was put in front of the IGV to obtain traces of axial velocity and measure the growth of rotating stall cells. Traverses performed in reverse regime demonstrated the overall uniformity of the flow during this part of the surge cycle.

Day reported the complete compressor map, surge trajectories for different sizes of the plenum and the respective cycle frequencies. The timing of deceleration, reversal and acceleration were also described as shown in figure 3.34 where the measurement of the absolute velocity during the surge can be observed.

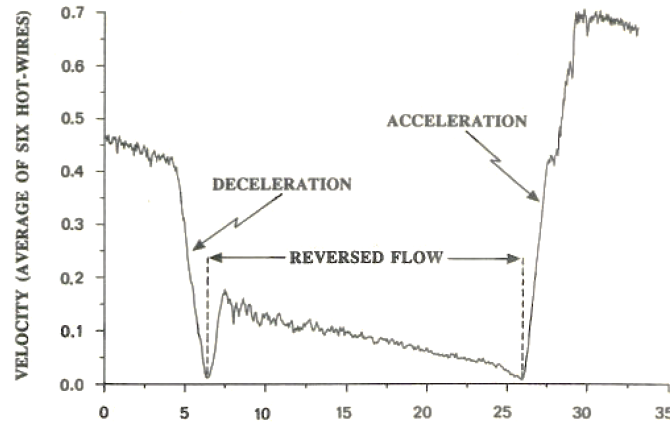


Fig. 3.34: Absolute velocity measured at compressor outlet during surge [15]

3.4 Summary

In this chapter is reported a literature review describing the various aspects of compressor stall. The first part of the review is the description of the current knowledge on the two main type of compressor stalls and how their performance can be characterized. A review of the models and methods existing to simulate these stall is also performed, ranging from lumped zero-dimensional codes to state-of-the-art three-dimensional CFD. Finally a description is offered of the typical set-up of

an experimental rig used to study compressor stall and what data can be obtained through its measures.

The methods to simulate compressor stall available in the literature have been assessed to see how they meet the requirements for a new tool as described in the research aim of this thesis, section 1.3.

- Three-dimensional high-fidelity-domain CFD offers the most detailed and accurate representation of the flow-field, however the examples reported in literature show computational times of weeks (or months) for a few rotor revolutions for a limited number of blade rows.
- Lumped models can be used to model surge and rotating stall, however they are not predictive and require the knowledge of the compressor performance in every regime.
- Through-flow codes can offer a two or three-dimensional representation of compressor stall with a lower computational cost; they solve the Euler equation and require empirical correlations to model the blade rows performance during every flow regime.

Based on this review the new code has been chosen to be a three-dimensional through-flow code using the body force method, empirical correlation and a new treatment for reverse flow; the details of this choice are described in section 3.2.5. This code will have to be able to model reverse flow, rotating stall and surge on low speed compressor rigs and be validated against experimental data wherever possible.

Such a code is not available in the public domain, the existing codes have not been validated or cannot be easily reproduced; a gap in the knowledge is then identified, which will be addressed in this research project.

Chapter 4

Methodology

4.1 Tool development overview

The tool methodology is shaped from the assumptions made at the project start. These were identified based on what was learnt from the literature review and which code capabilities are targeted. The code is required to have the following characteristics:

- solve the 3D unsteady, cylindrical, compressible Euler equations. No shear stress is modelled and friction is introduced only as a blade effect;
- the body force method is employed and the blade rows are replaced by a 3D force-field. The forces introduce on the flow the turning and the pressure loss imposed by the blades;
- turning and pressure loss are estimated using empirical correlations.

In this chapter the methodology of the code is described, expanding from these three initial assumptions to the details of the full working code. The Euler solver, body force method and empirical correlations are described in detail in sections 4.2 to 4.4. The successive two sections report the correlations used for forward flow and stall detection. Section 4.7 describes the novel method developed to treat reverse/separated flows. Section 4.8 describes the boundary conditions, the plenum/throttle model and their implementation in the code. Section 4.9 illustrates the inputs and outputs of the code and the methodology developed to efficiently create the complete compressor map with the tool. Section 4.10 describes briefly how the programme was coded and parallelised for higher computational speed.

4.2 Euler solver

In this section the Euler solver which has been developed for the code is described. The first section is devoted to explain the theory behind the choice of the scheme. The solver chosen is then described in section 4.2.2. A report of how the solver is implemented and some auxiliary algorithms employed is given in the last sections. The detailed implementation of the equations is provided in appendix A.

4.2.1 Monotone and TVD scheme

When solving the Euler equations in applications which are expected to contain flow discontinuities (e.g. shocks or pressure waves) it becomes important to consider some specific characteristics of the scheme employed [59]. A scheme can be:

- Monotonic
- Total Variation Diminishing
- Monotonicity preserving

A numerical scheme applied to a conservation equation of the form $\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}}{\partial x} = 0$ can be written in the form:

$$\mathbf{U}_i^{t+dt} = H(\mathbf{U}_{i-k}^t, \mathbf{U}_{i-k+1}^t, \dots, \mathbf{U}_{i+k}^t) \quad (4.1)$$

where the apex t indicates the time step. A scheme is *monotonic* if the function H is a monotone increasing function of each of the arguments of position $i - k < j < i + k$:

$$\frac{\partial H}{\partial \mathbf{U}_j} \geq 0 \quad (4.2)$$

In practical terms a monotone scheme has the property that:

- No new local maximum/minimum is created
- The value of the local maximum does not increase and the value of the local minimum does not decrease

This means that unlike many classic CFD methods (e.g. a classic upwind method) a monotone scheme does not introduce spurious/numerical oscillations when discontinuities are present. This behaviours are compared in figure 4.1.

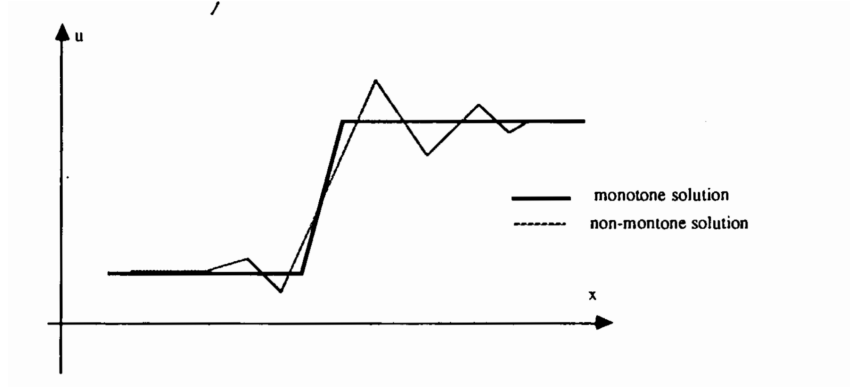


Fig. 4.1: Monotonic behaviour of a numerical scheme [59].

Another property is that a converged solution of a monotonic scheme is always a physically acceptable solution; i. e. such scheme can not produce any unphysical behaviour like a decrease in entropy through a shock (called rarefaction shock). This property is in fact known as *entropy condition*.

A scheme is *Total Variation Diminishing* or TVD when the integral of the conservative variable derivative on all the domain (Total Variation, TV) is always decreasing in time:

$$TV = \int \frac{\partial U}{\partial x} dx \quad (4.3)$$

$$TV(U^{t+dt}) \leq TV(U^t) \quad (4.4)$$

Finally the *monotonicity preserving* propriety means that, if the scheme was monotone at time t then it is monotone also at the next step $t + dt$.

The relationship between these characteristics, for a non-linear scheme, is:

- A monotone scheme is TVD
- A TVD scheme is monotonicity preserving

Godunov proved that a scheme can not be monotonic if it is more than first-order accurate, this is known as Godunov's order barrier theorem (Godunov, 1959; cited

by Hirsch [59]). While second or higher order solvers can not be monotonic they can be made TVD by using algorithms called *slope limiters*. These are algorithms which limit the derivatives used for interpolations, preventing them from overshooting and creating spurious oscillations. High order methods also do not naturally satisfy the entropy condition and additional algorithms are necessary to avoid un-physical solutions.

The properties of a monotone scheme are important when modelling compressor instabilities where we expect to see many pressure waves travelling across the domain and controlling the event dynamics. To accurately model a discontinuity with a classic CFD method it would be necessary to use a very fine grid which would increase the computational time, defeating the purpose of using a through-flow code. For this particular application it is very attractive to use a monotonic (or monotonicity preserving) scheme.

4.2.2 The Godunov scheme

The scheme selected to create the tool's solver is the first order monotone scheme proposed by Godunov, which wields his name [52]. It is a finite volume scheme where the inter-volume fluxes are based on the solution of the local Riemann problem.

Every time step of the Godunov scheme involves 3 separate stages:

1. The variables inside of the volumes are interpolated using a piecewise constant distribution to obtain the states on each side of the inter-volume boundaries.

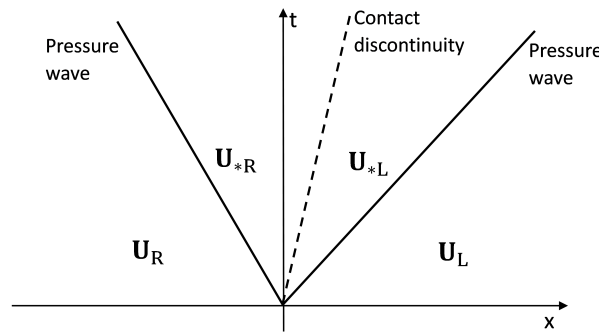


Fig. 4.2: Riemann problem at an inter-volume boundary.

2. At volume boundaries we can observe a Riemann problem as shown in figure 4.2. In the image the two states, left and right of the boundary, form a discontinuity on the edge (position $x = 0$) at the beginning of the time step ($t = 0$). Continuity and pressure waves are generated and start travelling from the origin point on the boundary. The structure shown in the figure is only one of the 10 possible combination of waves.

The HLLC (Harten-Lax-van Leer-Contact) Riemann solver is used to model an approximation of these waves and of the states between them. The state in which lies the line departing vertically from the starting point is the state in which the boundary remains during the time step. The fluxes calculated from this state are the physically correct one.

3. The fluxes calculated are put in the Euler equations to determine the flow-field at the next time step.

The formulations of the Godunov scheme and of the HLLC Riemann solver implemented in the code are taken from the book ‘Riemann Solvers and Numerical Methods for Fluid Dynamics’ of E. Toro [52].

The scheme is applied in the 3D code as an *unsplit* finite volume method, which means that no dimensional splitting is used and the solution is advanced in time using all fluxes in a single step. More complex methods exist for 3D applications where fluxes in different directions are used at different stages of a single time step [52]. These split techniques takes into account the interaction of the fluxes and are more accurate, however they increase significantly the computational time.

The solver calculates the optimal time step dt at every iteration based on the CFL (Courant-Friedrichs-Lewy) condition. The CFL condition for a Godunov scheme is set to a value of 0.5 to avoid interaction of waves coming from different boundaries inside of the volumes, as shown in figure 4.3. The time step is limited as:

$$dt \leq \frac{1/2dx}{a_{max}} \quad (4.5)$$

The basic form of the Godunov scheme here described is first order accurate in space. It can be transformed into a high resolution scheme by replacing in the

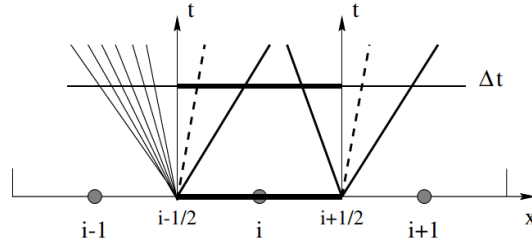


Fig. 4.3: Interaction of waves from different boundaries [52].

first stage the piecewise interpolation with a linear interpolation. The scheme then becomes the second order method known as MUSCL scheme (Monotonic Upwind Scheme for Conservation Laws) [52]. The MUSCL scheme is not naturally monotonicity preserving or TVD, but it can be made so by using *slope limiters*.

The monotonicity of the Godunov scheme, together with the conservative properties of a finite volume method, grants much more robustness when compared to the McCormack scheme previously employed in FENICE and in the early iterations of this code. Artificial viscosity had previously been a problem as it was in conflict with the high gradients typical of this type of simulation. The Godunov scheme does not require the use of artificial viscosity and no error is introduced because of it.

As a first order method the Godunov scheme introduces substantial diffusivity. However it is suggested that it could be advantageous to simulate compressor stall with Euler equations using a diffusive method. Since shear stress is absent, the presence of some numerical viscosity could prevent un-physical solutions with extreme recirculation from being modelled and overall it would stabilize the simulation.

The time stepping technique chosen to advance the solution in time is the Euler forward scheme, very simply:

$$\mathbf{U}^{t+dt} = \mathbf{U}^t + \frac{\partial \mathbf{U}}{\partial t} \cdot dt \quad (4.6)$$

The solver is then first order accurate in time and space.

4.2.3 Grid generation

Grid generation takes as input the annulus geometry and the number of axial, radial and circumferential elements N_x, N_r, N_θ . Axial and circumferential directions are discretised uniformly, the annulus span is divided at every axial position in uniform radial intervals. The domain is then divided in $N_x \times N_r \times N_\theta$ volumes. Figure 4.4 shows the volume shape in the meridional plane with their centroids and vertexes. The volumes, vertexes, centroid and any other geometrical feature are identified using a triplets of indices i, j, k , corresponding to the axial, radial and tangential positions.

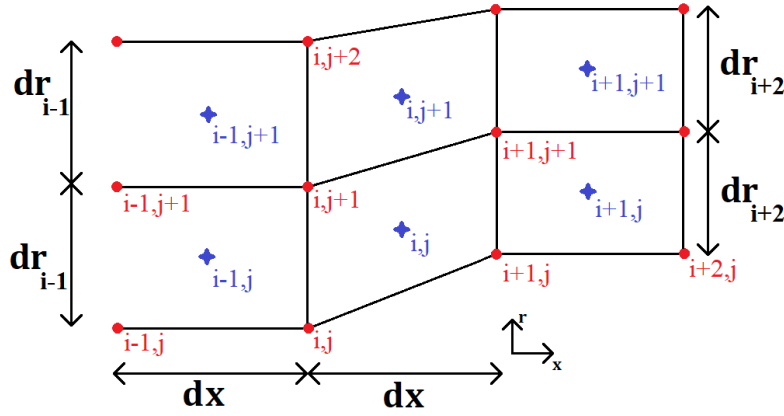


Fig. 4.4: Volumes' vertexes (red) and centroids (blue) with the indexing used.

Axial intervals dx and sector angles $d\theta$ are constant for every volume, the radial interval instead depends on the axial position (or axial index i), $dr = dr(i)$. The grid and all its geometric quantities are constant during the simulation and are pre-calculated.

Once the blade geometry is generated it is over-imposed on the domain. The volumes are divided into *rotor*, *stator* and *duct* elements depending if they are in the force-field of the blades or not, as in figure 4.5.

4.2.4 Equations

The equations solved in the code are the cylindrical, compressible, unsteady Euler equations:

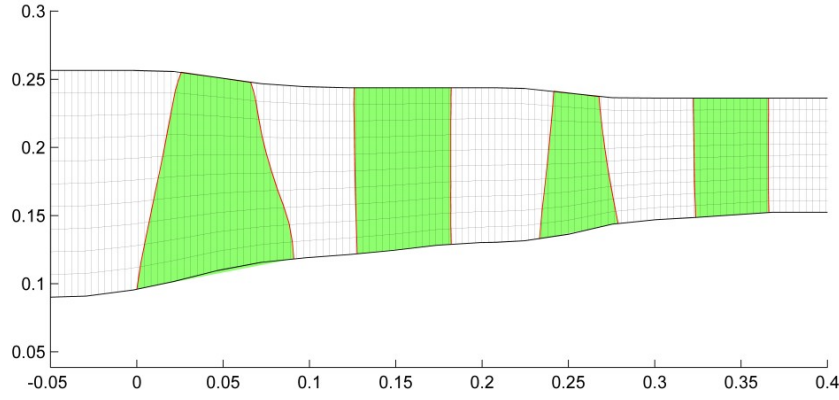


Fig. 4.5: Example of grid generation for the compressor rig described in NASA TP1493 [61].

$$\frac{\partial}{\partial t} \begin{bmatrix} \rho \\ \rho u_r \\ \rho u_\theta \\ \rho u_x \\ \rho E_0 \end{bmatrix} = -\frac{1}{r} \frac{\partial}{\partial r} \begin{bmatrix} \rho r u_r \\ \rho r u_r^2 \\ \rho r u_\theta u_r \\ \rho r u_x u_r \\ \rho r H_0 u_r \end{bmatrix} - \frac{1}{r} \frac{\partial}{\partial \theta} \begin{bmatrix} \rho u_\theta \\ \rho u_r u_\theta \\ \rho u_\theta^2 + P \\ \rho u_x u_\theta \\ \rho H_0 u_\theta \end{bmatrix} - \frac{\partial}{\partial x} \begin{bmatrix} \rho u_x \\ \rho u_r u_x \\ \rho u_\theta u_x \\ \rho u_x^2 + P \\ \rho H_0 u_x \end{bmatrix} + \begin{bmatrix} 0 \\ \frac{\rho u_\theta^2}{r} + \frac{\partial P}{\partial r} \\ -\frac{\rho u_\theta u_r}{r} \\ 0 \\ 0 \end{bmatrix} \quad (4.7)$$

Where the specific internal energy and enthalpy are calculated considering a calorically perfect gas:

$$E_0 = C_V \cdot T + \frac{1}{2}(u_r^2 + u_\theta^2 + u_x^2); \quad H_0 = C_P \cdot T + \frac{1}{2}(u_r^2 + u_\theta^2 + u_x^2) \quad (4.8)$$

The Euler equations are closed with the perfect gas equation:

$$P = \rho R T \quad (4.9)$$

The specific heat ratio is constant and is assumed to be $\gamma = 1.4$, the gas constant for air is taken as $R = 287 \text{ J/(kgK)}$. The specific heats at constant pressure and volume are therefore $C_P = 1004 \text{ J/(kgK)}$ and $C_V = 717 \text{ J/(kgK)}$.

Note that the primitive variables are:

$$\mathbf{V} = \begin{bmatrix} \rho \\ u_r \\ u_\theta \\ u_x \\ T \end{bmatrix} \quad (4.10)$$

The equations can be written concisely as:

$$\frac{\partial}{\partial t} [\mathbf{U}] = -\frac{1}{r} \frac{\partial}{\partial r} [\mathbf{E} \cdot r] - \frac{1}{r} \frac{\partial}{\partial \theta} [\mathbf{F}] - \frac{\partial}{\partial x} [\mathbf{G}] + [\mathbf{S}] \quad (4.11)$$

where U is the conservative variables vector, E, F, G are the radial, circumferential and axial fluxes and S is the source terms vector.

The effect of the blades is introduced using the body force method which in the equations is represented by a forcefield $\mathbf{f}$ added to the source terms. This consists of three forces in the three directions and a work exchange term W_{EX} added only in the rotors.

$$\mathbf{f} = \begin{bmatrix} 0 \\ F_r \\ F_\theta \\ F_x \\ W_{EX} \end{bmatrix} \quad (4.12)$$

The equations are also modified inside of the blade passages by removing the circumferential flux, which is blocked by the presence of the blades .

For the points inside of the stators the equations become:

$$\frac{\partial}{\partial t} \begin{bmatrix} \rho \\ \rho u_r \\ \rho u_\theta \\ \rho u_x \\ \rho E_0 \end{bmatrix} = -\frac{1}{r} \frac{\partial}{\partial r} \begin{bmatrix} \rho r u_r \\ \rho r u_r^2 \\ \rho r u_\theta u_r \\ \rho r u_x u_r \\ \rho r H_0 u_r \end{bmatrix} - \frac{\partial}{\partial x} \begin{bmatrix} \rho u_x \\ \rho u_r u_x \\ \rho u_\theta u_x \\ \rho u_x^2 + P \\ \rho H_0 u_x \end{bmatrix} + \begin{bmatrix} 0 \\ \frac{\rho u_\theta^2}{r} + \frac{\partial P}{\partial r} + F_r \\ -\frac{\rho u_\theta u_r}{r} + F_\theta \\ F_x \\ 0 \end{bmatrix} \quad (4.13)$$

For the points inside of the rotors the equations are:

$$\frac{\partial}{\partial t} \begin{bmatrix} \rho \\ \rho u_r \\ \rho u_\theta \\ \rho u_x \\ \rho E_0 \end{bmatrix} = -\frac{1}{r} \frac{\partial}{\partial r} \begin{bmatrix} \rho r u_r \\ \rho r u_r^2 \\ \rho r u_\theta u_r \\ \rho r u_x u_r \\ \rho r H_0 u_r \end{bmatrix} - \frac{\Omega}{r} \frac{\partial}{\partial \theta} \begin{bmatrix} \rho \\ \rho u_r \\ \rho u_\theta \\ \rho u_x \\ \rho E_0 \end{bmatrix} - \frac{\partial}{\partial x} \begin{bmatrix} \rho u_x \\ \rho u_r u_x \\ \rho u_\theta u_x \\ \rho u_x^2 + P \\ \rho H_0 u_x \end{bmatrix} + \begin{bmatrix} 0 \\ \frac{\rho u_\theta^2}{r} + \frac{\partial P}{\partial r} + F_r \\ -\frac{\rho u_\theta u_r}{r} + F_\theta \\ F_x \\ W_{EX} \end{bmatrix} \quad (4.14)$$

In equations 4.14 the circumferential flux is substituted with a term which introduces the translation of the conservative variables with the blade passages. This term was initially proposed by Gong [43]. This term is solved using a 3rd order upwind method in order to reduce the dispersion due to the low number of elements in the circumferential direction. By taking into account the rotation with this term, the rotor equations can be solved in the absolute frame of reference with no need for a multi-domain approach.

Also the equation in the blade passages can be written in a concise form as:

$$\frac{\partial}{\partial t} [\mathbf{U}] = -\frac{1}{r} \frac{\partial}{\partial r} [\mathbf{E} \cdot r] - \frac{\Omega}{r} \frac{\partial}{\partial \theta} [\mathbf{U}] - \frac{\partial}{\partial x} [\mathbf{G}] + [\mathbf{S}] + [\mathbf{f}] \quad (4.15)$$

where Ω is 0 for the stators.

4.2.5 Solver implementation

This subsection details how the finite volume Godunov scheme is practically implemented in the present code.

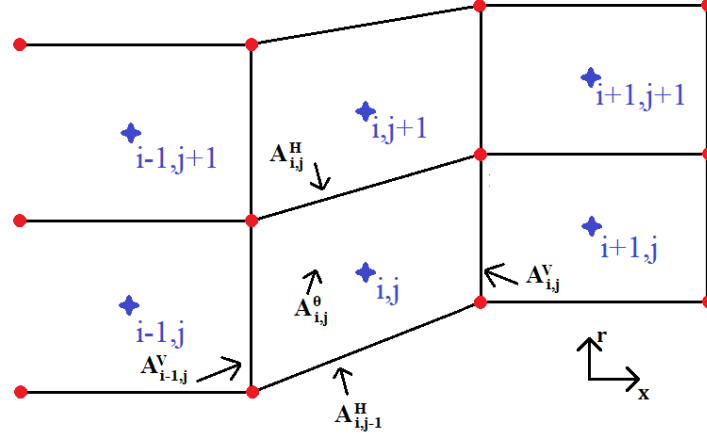


Fig. 4.6: Areas nomenclature for a generic grid element.

The nomenclature of the areas in the grid is shown in figure 4.6. The areas forming the volume i, j, k are two vertical A^V , two horizontal A^H (although in general these are not horizontal) and two circumferential A^{θ} . The circumferential areas are superimposed in the image and they are positioned at $\theta_k \pm d\theta/2$.

During every time step the fluxes across all of the interface areas are calculated. Hereafter the calculation of the flux across $A_{i,j}^H$ is used as example.

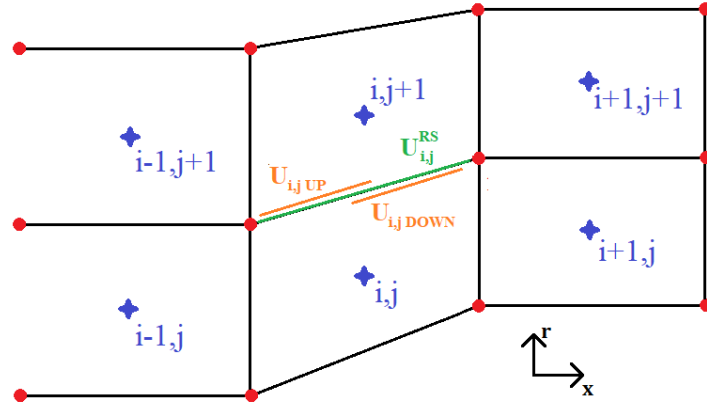


Fig. 4.7: Flux calculation across an horizontal area.

The first (projection) step in the Godunov scheme is the piecewise interpolation. The primitive variables at the volumes' interface ($\mathbf{V}_{i,j\ UP}$, $\mathbf{V}_{i,j\ DOWN}$) are simply those stored in the volume centroids, $\mathbf{V}_{i,j}$ and $\mathbf{V}_{i,j+1}$. While ρ , T , P and u_θ are directly used, radial and axial velocities are projected into the normal and parallel direction to the wall, $u_\perp$ and $u_\parallel$.

The HLLC Riemann solver employed calculates on the interface an approximate value of the waves and of all the inter-wave states. The primitive variables at the interface $\mathbf{V}_{i,j}^{RS}$ are extracted from the Riemann problem solution and from these the flux $\mathbf{E}_{i,j}$ is calculated as in equation 4.7. The axial fluxes G across the areas A^H and the circumferential fluxes $\mathbf{F}$ across the area A^θ are calculated in a similar fashion.

The third step is the calculation of the time derivative. As this is a finite volume method the equations 4.7, 4.13 and 4.14 are modified by transforming the space derivatives into the integral of the flux through the volume area. This is calculated as the sum of the fluxes $\mathbf{E}, \mathbf{F}, \mathbf{G}$ through the 6 areas of the volume. The source terms $\mathbf{S}$ and $\mathbf{f}$ instead are integrated as constant in the element volume.

Special attention must be paid for the pressure. The pressures on areas A^V and A^θ are acting along only one of the cylindrical coordinates, and are included in the fluxes. However, the pressure on the interface A^H must be applied in both the radial and axial directions. The pressure is multiplied by the projection of the area A^H on these direction and introduced in both the equations.

The density equation, as it is implemented, is:

$$\begin{aligned} \frac{\partial \rho}{\partial t} \cdot V_{i,j} = & (\rho u_\perp)_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H - (\rho u_\perp)_{i,j,k}^{RS} \cdot A_{i,j}^H + \left((\rho u_\theta)_{i,j,k-1}^{RS} - (\rho u_\theta)_{i,j,k}^{RS} \right) \cdot A_{i,j}^\theta \\ & + (\rho u_x)_{i-1,j,k}^{RS} \cdot A_{i-1,j}^V - (\rho u_x)_{i,j,k}^{RS} \cdot A_{i,j}^V \quad (4.16) \end{aligned}$$

The implementation of all the equations is reported in appendix A.

Once the derivative in time are calculated the solution is advanced to the next time step using equation 4.6.

4.3 Body force method

In this section the body force method used is described. The version detailed here is valid for the unstalled forward flow only. Modifications are necessary to apply body forces to separated/reversed/mixed flow, these are explained in section 4.7.

As suggested by Longley [41] the length-scale of a rotating stall cell is greater than a few blade pitches. When modelling rotating stall it is then sufficient to model only a reduced number of virtual passages in place of the physical ones. The virtual passages are the circumferential elements of the grid, with the number of elements in the order of the number of real blades. There is then no resolution of the flow profile inside the channels, instead the flow inside each grid element is a pitchwise-average of the flow inside one channel or more. The virtual passages are not capable to model features of a single passage (like for example a particular passage stalling) but should instead be seen as a statistical average of a large number of blades contained in a single circumferential element. This is the so-called *infinite number of blades* assumption. Since the number of virtual passages is constant along the compressor, the ratio of real/virtual passages varies between compressor stages and can be greater or less than unity (see figure 4.8).

Since the fluxes in the circumferential direction are set to zero for blade rows, this means that the flow entering a stator row at a position k exits at the same position k and same angle θ . The stagger angle of the rows is ignored and the flow is artificially shifted in the θ direction while passing across each blade row. No correction was introduced as this does not produce any significant error in the model.

4.3.1 Pseudo-meridional planes

The body forces are introduced in every element inside the blade rows along the so called *pseudo-meridional planes*. These planes are formed by all the volume centroids with the same radial index j and as such are not necessarily flat but follow the shape of the grid. An example of these planes is shown in figure 4.9. The axial and radial velocities of the volumes inside of the blade rows are projected on the pseudo-planes, to obtain a velocity parallel to the plane u_ξ as in figure 4.10. As

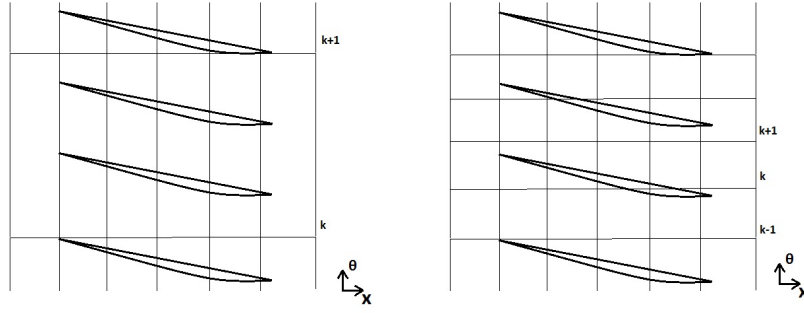


Fig. 4.8: Real and virtual blade passages in the tool. Their ratio can be higher or lower than 1.

shown in this figure the local direction of the plane λ is given by the local shape of the grid, in particular:

$$\lambda_{i,j} = \tan^{-1} \left(\frac{r_{i+1,j} - r_{i-1,j}}{2 \cdot dx} \right) \quad (4.17)$$

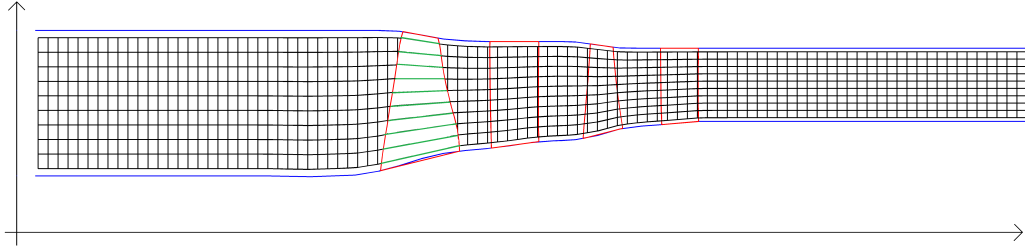


Fig. 4.9: Meridional view of the pseudo-plane in a rotor blade row. NB The grid in the picture is formed by volume centroids, not vertexes.

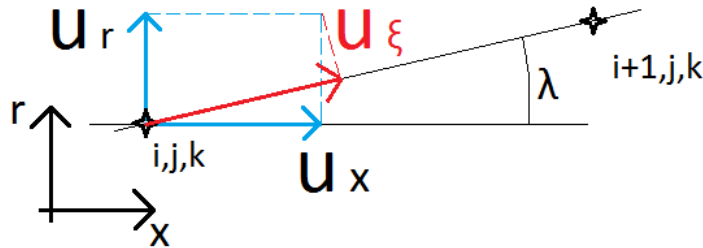


Fig. 4.10: Axial and radial flow projection on the local pseudo-plane.

4.3.2 Regimes

During compressor stall, different types of *flow regimes* are expected to occur, these are shown in figure 4.11.

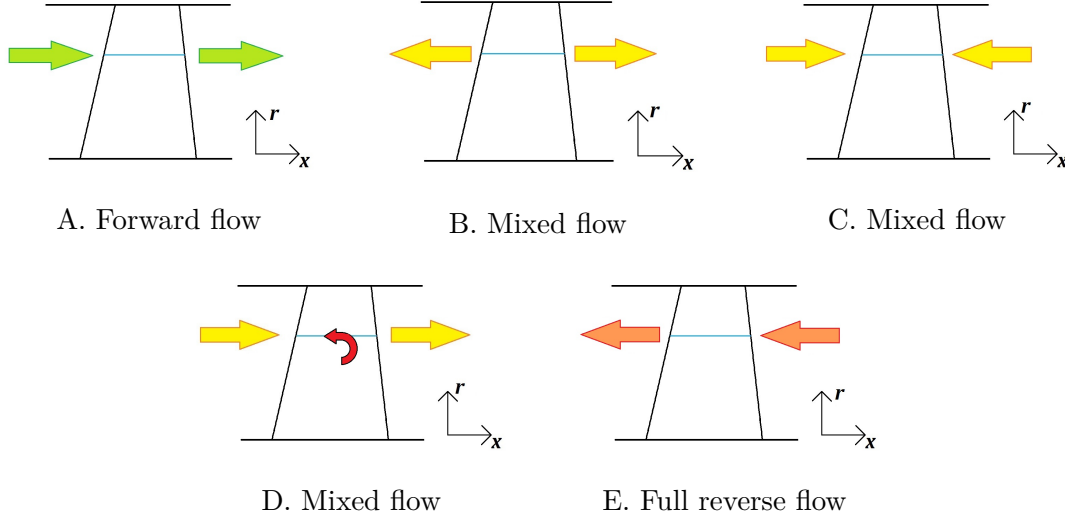


Fig. 4.11: Possible flow regimes.

The flow regime is evaluated at every pseudo-meridional plane j , at every circumferential position k , at every blade row. The regime is forward flow if in every point on the pseudo-plane $u_\xi > 0$, it is reverse flow if in every point $u_\xi < 0$, it is mixed flow in every other case. The regime evaluation is performed at the beginning of every time step.

4.3.3 Direction imposed

The body forces impose a direction β along the pseudo-meridional plane. This is calculated starting from the interpolation of the blade camberline on the pseudo-meridional plane, β_0 . Two modifications are applied to the camberline β_0 :

- At the leading edge it is modified to match the incoming flow angle α_{IN} . This allows to keep the body forces parallel and perpendicular to the flow.
- At the trailing edge it is modified to match the estimated exit angle and introduce the correct turning in the flow.

The exit angle is obtained from the deviation δ which is defined as the difference between the flow angle and the blade metal angle at outlet

$$\delta = \alpha_{OUT} - \beta_{OUT}. \quad (4.18)$$

The deviation δ is estimated by the empirical correlations.

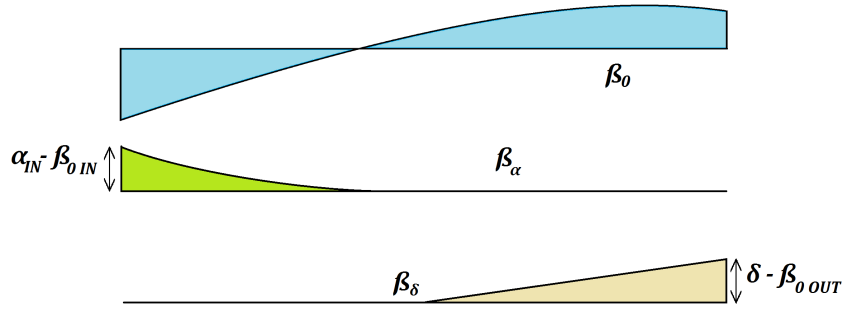


Fig. 4.12: Components of the imposed direction β .

Shape functions are used to gradually modify the direction imposed and obtain smooth direction changes. An example of two shape functions is shown in figure 4.12. The shape function β_α used to match the inlet angle has a value of 1 at the leading edge and decreases to 0 in the first half of the chord. The shape function β_δ used to introduce the deviation instead grows from 0 at mid-chord to 1 at the trailing edge. In the tool their shape and length can be set by the user.

Finally the direction imposed is :

$$\beta = \beta_0 + (\alpha_{IN} - \beta_{0\,IN}) \cdot \beta_\alpha + (\delta - \beta_{0\,OUT}) \cdot \beta_\delta \quad (4.19)$$

In case of reverse flow, only the deviation is added at the flow outlet (which in reverse flow is at the blade leading edge). For blades operating in mixed regimes the direction imposed is purely the camberline β_0 . During a flow reversal the instantaneous flow direction at the blade inlet can become very close to 90° if the axial velocity goes to zero. In this case the direction imposed β at the inlet would also become close to 90° . However the body force method does not have a solution for

purely circumferential flows and becomes unstable at angles very close to 90° . A similar problem would occur if during reverse flow the angle after mixing would be estimated to be $> 85^\circ$ and the body force would try to impose that. To avoid any of these situations the imposed direction is limited to a maximum value of 80° in any point of the domain.

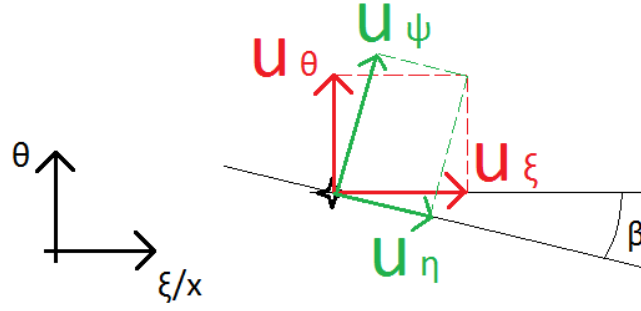


Fig. 4.13:]
Flow velocity decomposition on the imposed direction.

The velocity u_ξ and the circumferential velocity u_θ are projected on the direction β and divided into components normal u_ψ and parallel u_η , used in the body force calculation.

4.3.4 Body force formulation

The effects of the blades on the flow is introduced by the body forces. Two different effects are introduced independently by two force terms:

- A turning force F_N turns the flow in the direction imposed by the passage without changing the total pressure in the relative frame of reference. This force is normal to the flow in the relative frame of reference.
- A pressure loss force F_{PL} imposes locally the total pressure loss estimated by the empirical correlation, ΔP_0 . This force is always parallel to the flow in the relative frame of reference.

The parallel force F_{PL} is in the same direction of u_η , while the normal force F_N is in the direction of u_ψ . These are projected in the θ and ξ direction; F_ξ is further

projected in the x and r directions. The forces F_x , F_r and F_θ obtained are then introduced in the Euler equation as source terms. The work exchange term W_{EX} is calculated multiplying F_θ and the blade velocity.

$$W_{EX\ i,j} = F_{\theta\ i,j} \cdot \Omega \cdot r_{i,j} \quad (4.20)$$

In rotor blade rows the forces, which are aligned to the flow in the relative FOR, are not aligned in the absolute one. The turning force can then increase the total pressure of the flow and produce a total pressure rise. Similarly the rotors can extract total pressure when the compressor is windmilling.

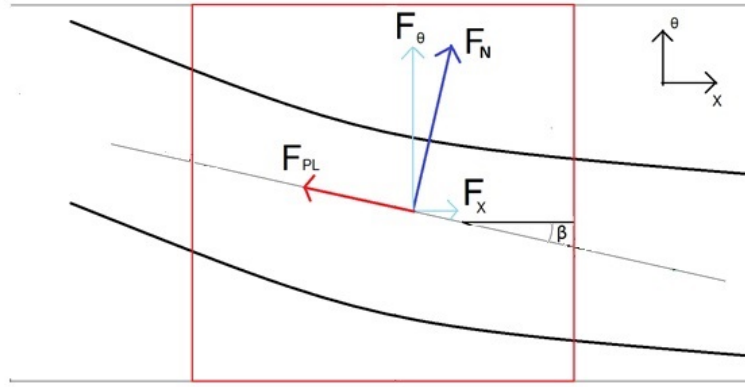


Fig. 4.14: Direction of turning and pressure loss forces in the relative FOR; in red the control volume considered.

The value of the turning force was taken from what was suggested by Longley [41]:

$$F_{Ni} = \frac{\rho_i u_{\eta i}^2}{R_{CURi}} + F_{CORi} \quad (4.21)$$

The first element takes into account the change in momentum along the local radius of curvature of the blade passage as a centripetal force. The radius of curvature is obtained by modelling the local camberline of the blade as a circular arc.

$$R_{CURi} = \frac{\tan(\beta_{i+1}) + \tan(\beta_{i-1})}{2} \cdot \left(1 + \left(\frac{\tan(\beta_{i+1}) + 2\tan(\beta_i) + \tan(\beta_{i-1})}{2} \right)^2 \right)^{-1.5} \quad (4.22)$$

The component F_{COR} is a corrective force which reacts to the misalignment between the direction imposed β and the actual flow direction. It is proportional to the velocity u_ψ :

$$F_{CORi} = -\rho_i |u_{\xi i}| \frac{u_{\psi i}}{s \cos \beta_i} \cdot K_{COR} \quad (4.23)$$

This force is essential to model the response of the blade surfaces to the flow during transients and it also ensures that in the case of strong radial redistribution in the blade, the correct flow direction will be maintained. The parameter K_{COR} is a constant value which controls the magnitude of the blade response (and thus the blade stiffness). Longley proposed a value of 2 to guarantee that the flow will exit the passage aligned with the stagger angle in reverse flow. However, in the test cases studied by this author the value was found to be insufficient for high speed and high swirl simulations, instead a value of 10 was used for K_{COR} . The formulation by Longley is also slightly different from what used in equation 4.23, which justifies the re-tuning of the constant.

The pressure loss force F_{PL} is calculated based on an approximation proposed by Longley which gives accurate results in a wide range of flow velocities:

$$F_{PL} = -\Delta P_{0i} \frac{P_i/P_{0i}}{\Delta x \cdot \cos \beta} \quad (4.24)$$

The turning force F_N presented in equation 4.23 does not take into account the pressure gradient inside of the channels. The formulation based only on the curvature in fact becomes more and more under-predicting as Mach number increases. Gong proposed that a force proportional to the local axial gradient of pressure should be included [43]:

$$F_{PRESSURE} \propto \frac{\partial P}{\partial x} \cdot \sin \beta_i \quad (4.25)$$

The local pressure gradient is a function of the geometry, the pressure loss and the flow-field. The derivation was performed by Brand [44] starting from the differential version of the Euler equation. The final version used in the code is:

$$F_{Ni} = \frac{(1 - M_\xi^2)}{\cos^2(\beta)(1 - M^2) + K_{OFF}} \cdot \frac{\rho_i u_{\eta i}^2}{R_{CURi}} + \frac{\tan \beta (1 + (\gamma - 1)M^2)}{1 - M^2 + K_{OFF}} F_{PL} + F_{COR} \quad (4.26)$$

The constant offset K_{OFF} is added to avoid singularities when the Mach number approaches 1. While Brand proposed a constant value in the present work K_{OFF} is set to zero for Mach numbers up to 0.9 and is introduced gradually at higher values. This guarantees avoiding the singularity while retaining higher accuracy at lower M.

4.4 Empirical correlations

The body force model described in section 4.3 requires the knowledge of the total pressure loss and deviation at every pseudo-plane. These are quantified by means of two parameters, the deviation δ and the pressure loss coefficient ω . The coefficient ω is defined as the total pressure loss over the relative inlet dynamic head:

$$\omega = \frac{P_{0OUT} - P_{0IN}}{P_{0IN} - P_{IN}} \quad (4.27)$$

The IN and OUT subscript refers to the position where the flow effectively enters or exits the blade. These position are reversed when the flow is reversed. The coefficient ω has no meaning when inlet and outlet can not be clearly defined.

At every time step, ω is estimated using the empirical correlation and then multiplied by the inlet dynamic head to obtain the overall total pressure loss. The pressure loss is redistributed along the axial chord using a sinusoidal distribution, allocating for every grid element inside the pseudo-plane a local pressure loss, ΔP_0 . In every element is then calculated a force F_{PL} to introduce this pressure loss.

The values of ω and δ are calculated using correlations which are developed for steady state flows. These are extended for transients application using a first order time-lag law:

$$\omega^t = \omega^{t-dt} + (\omega^{t-dt} - \omega_{SS}^t) \cdot \tau/dt. \quad (4.28)$$

ω_{SS} (or δ_{SS}) are the steady state values calculated at the time step t , the previous

time step being $t - dt$. τ is the time constant, calculated as the time for the flow to cross the blade row:

$$\tau = c_x / u_x \quad (4.29)$$

When the change in ω and δ is due to the movement of the operating point in the unstalled region of the map, this constant is small compared to the overall timing of the event. However this delay becomes important in the modelling of the passage stalling when both parameters change substantially in a short amount of time. When a passage stalls the change in deviation and pressure loss becomes effective when the separated vortex which is started at the leading edge reaches the trailing edge (see figure 4.15). This particular definition of τ is chosen to model this event because the speed of the vortex shedding is the flow velocity itself.

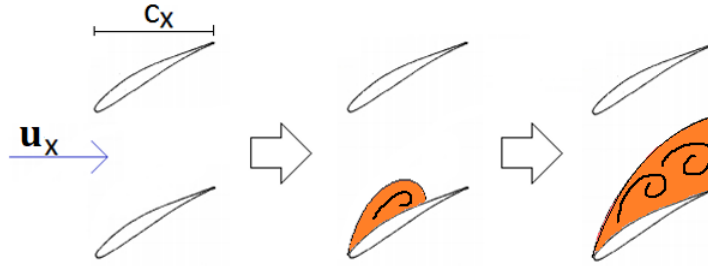


Fig. 4.15: Separated cell shed from the leading edge during passage stalling.

The values of deviation and pressure loss coefficient calculated at every time step are specific of a particular *virtual* blade passage (which represent one or more blade passages) and is stored at a particular radial and tangential position j, k . If the passage is part of a rotor blade row this will rotate in time and the values of ω and δ must be translating together with it (figure 4.16). In order to take into account this effect at every time step the coefficients are rotated using a 3rd upwind method. The formulation used to model the rotation from time t to $t + 1$ of the pressure loss coefficient calculated at a circumferential position k is:

$$\begin{cases} \omega_k^{t+dt} = -\frac{\Omega}{6} \cdot (2\omega_{k+1}^t + 3\omega_k^t - 6\omega_{k-1}^t + \omega_{k-2}^t)/d\theta, & \Omega \geq 0 \\ \omega_k^{t+dt} = +\frac{\Omega}{6} \cdot (2\omega_{k-1}^t + 3\omega_k^t - 6\omega_{k+1}^t + \omega_{k+2}^t)/d\theta, & \Omega < 0 \end{cases} \quad (4.30)$$

The use of a 3rd order method reduces the smearing caused by the small number of circumferential sectors. In fact the size of the element $d\theta$ is normally an order of magnitude bigger than the rotation $\Omega \cdot dt$.

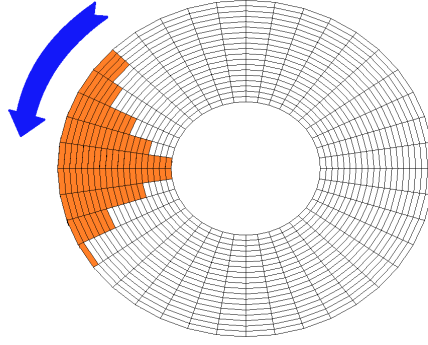


Fig. 4.16: Example of stalled virtual passages in a rotor blade row and their translation.

The order at which these operations are implemented is:

1. Steady state values of ω and δ at time t are calculated,
2. the rotation algorithm is applied, to model the shifting during the last time step $(t - dt) \rightarrow (t)$,
3. the rotated values of the coefficients are updated with the time lag law, to obtain the coefficients used in the present time step.

4.5 Forward flow correlations

The correlations described in this section are used for forward unstalled flow. These are mostly based on experimental measurements from cascades with typical blade profiles. All the correlations described here have been proposed in literature and reported by Aungier [60]. In the following section all the angles are in degrees, the nomenclature is shown in figure 4.17.

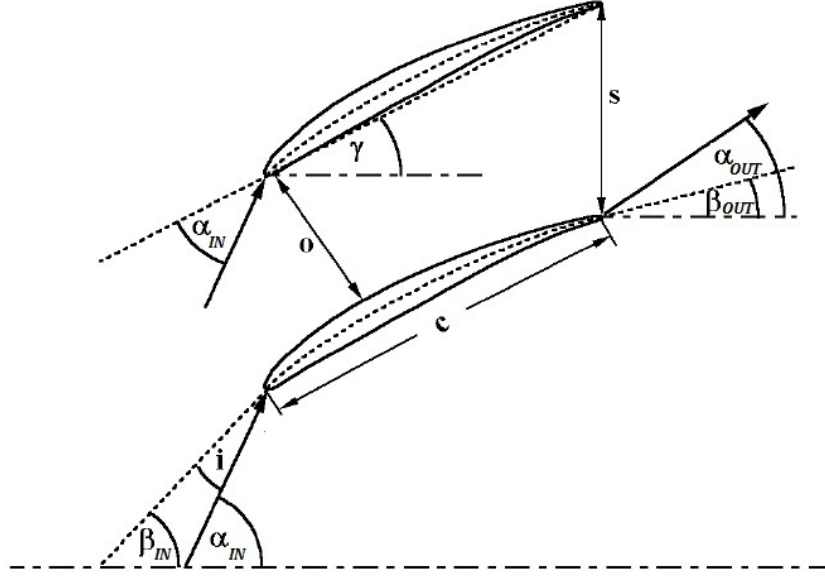


Fig. 4.17: Nomenclature of angles and dimensions in a blade passage [60].

4.5.1 Pressure loss coefficients

Before starting the simulation a design value of loss coefficient is calculated, which is function only of the geometry and therefore constant.

The design value of the aerodynamic angles α^* at inlet and outlet are required for this correlation. If these angles are not known specific correlations are available in the code to estimate them based on profile geometry.

To obtain the design loss coefficient the first step is calculating the design value of the diffusion factor as:

$$D^* = 1 - \frac{u_{IN}^*}{u_{OUT}^*} + \left[\frac{r_{OUT} u_{\theta OUT}^* - r_{IN} u_{\theta IN}^*}{(r_{OUT} + r_{IN}) \cdot \sigma \cdot u_{\xi IN}^*} \right] \quad (4.31)$$

All the velocities in equation 4.31 are values in design condition. The design profile loss coefficient is then calculated as:

$$\omega^* = [K_1(K_2 + A_1 D^* + A_2 D^{*4})] \frac{2\sigma}{\cos(\alpha_{OUT}^*)} \quad (4.32)$$

The constant K_1 , A_1 and A_2 should be obtained through data fitting if possible, but general value are proposed by Aungier. The constant K_2 takes into account the

end wall losses and is estimated using the formula:

$$K_2 = 1 + (s/h) \cos(\alpha_{OUT}^*) \quad (4.33)$$

In the code these constants have been obtained by data fitting values from NASA report TP1493 [61]. The data fitting process had already been performed during the development of SOCRATES and those values are imported in the code.

From this point onward the coefficients are not pre-calculated but computed at every time step. The attack angles λ at positive and negative stall are computed using empirical equations for low Mach number cascades:

$$\lambda_{NS} - \lambda^* = -9 + \left[1 + \left(\frac{30}{\alpha_{INS}} \right)^{0.48} \right] \cdot \frac{\theta}{4.176} \quad (4.34)$$

$$\lambda_S - \lambda^* = 10.3 + \left(2.92 - \frac{\alpha_{INS}}{15.6} \right) \cdot \frac{\theta}{8.2} \quad (4.35)$$

The attack angles and flow angles are one function of the other, $\alpha_{IN} = \lambda + \gamma$, therefore these equations are solved iteratively. Incidence angles i and stall margins R at stall conditions are calculated as:

$$R_{NS} = \lambda^* - \lambda_{NS} = i^* - i_{NS} \quad (4.36)$$

$$R_S = \lambda_S - \lambda^* = i_S - i^* \quad (4.37)$$

It is assumed that the value of the loss coefficient will be $2\omega^*$ at stall condition (positive and negative). The profile of the function $\omega = f(i)$ is assumed parabolic and passing through the 3 points at stall and design condition, as shown in figure 4.18.

For increasing Mach numbers the profile of loss coefficient is shifted upward and the angle of minimum loss changes. The applicability of the correlations previously

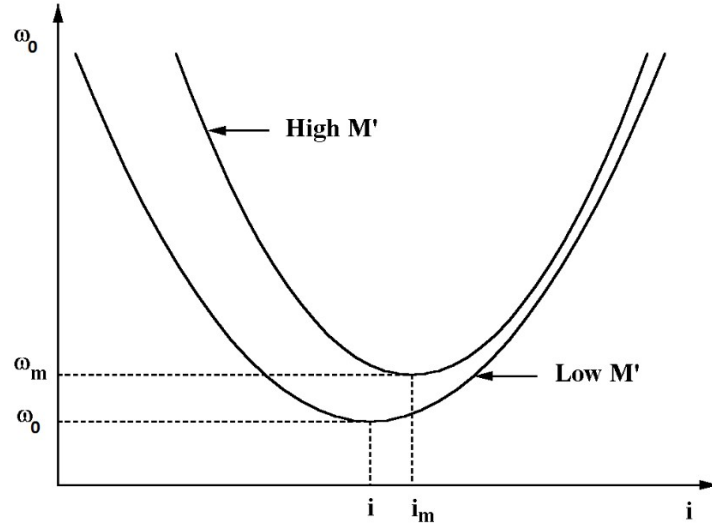


Fig. 4.18: Loss coefficient profile and the effect of Mach number [60].

shown can be expanded to high Mach number using the correction:

$$i_{NS} = \frac{i^* - R_{NS}}{1 + 0.5M_{IN}^3} \quad (4.38)$$

$$i_S = i^* - \frac{R_S}{1 + 0.5K_{SH}M_{IN}^3} \quad (4.39)$$

K_{SH} is a shape factor with value 1.0 for NACA profile, 1.1 for C4 series and 0.7 for double circular arc profiles.

The new angle of minimum loss and the pressure loss at that condition are defined as:

$$i_m = i_{NS} + ((i_S - i_{NS})R_{NS})/(R_{NS} + R_S) \quad (4.40)$$

$$\omega_m = \omega^* \cdot \left(1 + \frac{(i_m - i^*)^2}{R_S^2}\right) \quad (4.41)$$

The loss coefficient is then interpolated using a normalised value of the incidence angle, as shown in Figure 4.19.

The normalised angle, ξ is calculated as:

$$\xi = (i - i_m)/(i_S - i_m); \quad i \geq i_m \quad (4.42)$$

$$\xi = -(i - i_m)/(i_{NS} - i_m); \quad i < i_m \quad (4.43)$$

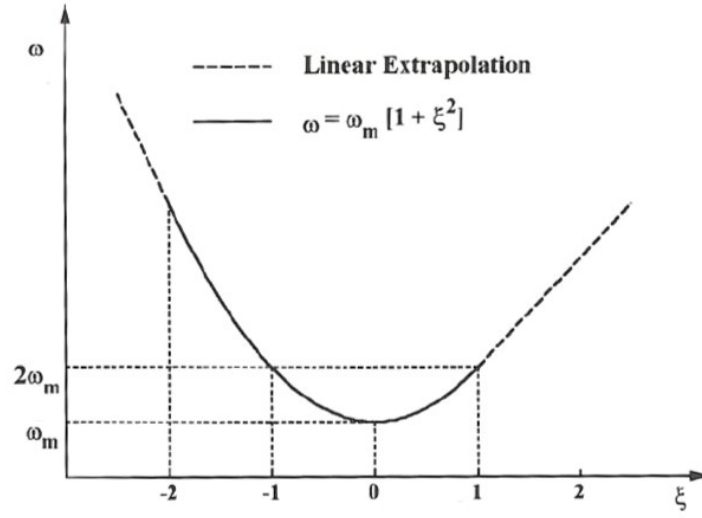


Fig. 4.19: Interpolation of pressure loss coefficient as proposed by Aungier [60].

The loss coefficient ω is interpolated considering again a parabolic profile, with the coefficient being the minimum at i_m and double the minimum value at the corrected values of negative and positive stall. Outside of the range $-2 < \xi < 1$ a linear extrapolation is used. For the cascade performances beyond negative stall this extrapolation is used as there are no better formulations in literature. Some correlations used for reverse flow might be adapted in the future to model this condition. To model the behaviour after positive stall instead a different set of correlations is used.

The correlations described in this section do not consider shock losses and are not suitable for transonic blades.

4.5.2 Deviation

The deviation is estimated using the procedure described by Aungier [60]. The first step is calculating the deviation for a zero camber blade in design conditions with the equation:

$$(\delta^*)_{1.0} = 0.01\sigma|\alpha_{IN}^*| + [0.74\sigma^{1.9} + 3\sigma] \cdot \left(\frac{|\alpha_{IN}^*|}{90} \right)^{(1.67+1.09\sigma)} \quad (4.44)$$

From this specific case a more general deviation for any blade in design condition

can be calculated as:

$$b = 0.9625 - 0.17 \frac{|\alpha_{IN}^*|}{100} - 0.85 \left(\frac{|\alpha_{IN}^*|}{100} \right)^3 \quad (4.45)$$

$$m = \frac{0.17 - 0.0333 \frac{|\alpha_{IN}^*|}{100} + 0.333 \frac{|\alpha_{IN}^*|}{100^2}}{\sigma^b} \quad (4.46)$$

$$\delta^* = K_{SH} K_{t,\delta} \cdot (\delta^*)_{1.0} + m \cdot \theta \quad (4.47)$$

The factor K_{SH} is the shape factor presented in the previous paragraph. The factor $K_{t,\delta}$ is a thickness correction factor calculated as:

$$K_{t,\delta} = 6.25 \cdot \frac{t_b}{c} + 37.5 \cdot \left(\frac{t_b}{c} \right)^2 \quad (4.48)$$

The deviation in off-design conditions is calculated as a function of incidence angle and meridional velocity ratio as:

$$\frac{\partial \delta}{\partial i} = \frac{1 + (\sigma + 0.25\sigma^4) \left(\frac{|\alpha_{OUT}^*|}{53} \right)^{2.5}}{e^{3.1\sigma}} \quad (4.49)$$

$$\delta = \delta^* + \frac{\partial \delta}{\partial i} (i - i^*) + 10 \left(1 - \frac{u_{\eta OUT}}{u_{\eta IN}} \right) \quad (4.50)$$

4.5.3 Rolls-Royce correlations

The forward flow correlations described from Aungier were found to poorly model the performance of the C106 rig, in particular the deviation. The current project is funded and industrially supervised by Rolls-Royce plc which is also the manufacturer of the C106 compressor rig blades. The company offered to the author to study the correlations used by the compressor design team in Rolls-Royce and import part of them in the program. The forward flow correlations for both δ and ω were transferred in the code and are available as an alternative to those in literature.

The original method is a mean-line calculation used during the initial stage of

the design. It calculates at every blade row one average value of δ and ω for the whole span. Unlike the forward flow correlations described previously this ω is a sum of a blade profile loss and an end-wall loss calculated separately.

This method has been adapted to use in 3D by applying the profile loss and the deviation correlations to all the pseudo-meridional planes and at all circumferential position. The end-wall losses are also calculated, but only once for every blade row (at every circumferential position). Although the end-wall losses are physically generated at hub and tip they can not be introduced in the domain only at the extreme positions. Since no shear stress is modelled, there is no momentum exchange between the streamlines in the flow. Therefore the losses at the end-wall will not mix span-wise downstream but they will concentrate on the first and last span-wise elements unrealistically. To avoid this problem the losses are introduced with a parabolic distribution in the blade row where they are generated (in a way they are "pre-mixed"). In practice the losses are estimated with the correlation and distributed along the span, then an end-wall loss coefficient ω_{EWj} is calculated and added to the blade profile loss.

These correlations are Rolls-Royce proprietary and commercially sensitive, therefore they will not be described any further here. In the scope of this thesis they are simply an alternative to the correlations reported in the previous section.

4.6 Stall detection

A criterion proposed by Aungier [60] is used to identify when a virtual blade passage stalls.

The equivalent relative velocity ratio v_{RE} is calculated from the total pressure in the actual flow field and represents the instantaneous diffusion of the blade passage. This is then compared with the velocity ratio at the peak value of diffusion, beyond which stall occurs (RHS of equation 4.51). This is calculated based only on the

blade geometry. Stall occurs if:

$$v_{RE} = \sqrt{\frac{P_{0OUT} - P_{OUT}}{P_{0IN} - P_{IN}}} < (2.2/D_{EQ})^{0.6} \cdot \frac{\frac{0.15 + 11t_b/c}{0.25 + 10t_b/c}}{1 + 0.4 \left[\frac{\theta\sigma}{2 \sin(\theta/2) \cos(\gamma)} \right]^{0.65}} \quad (4.51)$$

The first term of the RHS is a correction, function of the equivalent diffusion factor. It is added only when $D_{EQ} > 2.2$ as for high values of diffusion this criteria was found to be too conservative.

The δ and ω coefficients are calculated using the forward flow correlations until the stall is detected by this criteria. If $i > i_S$ in the calculation of the pressure loss coefficient then the ω is extrapolated from the "stalled" region, however the blade is not considered stalled unless this criterion (eq. 4.51) is met. When the stall is detected a different set of calculation is used to estimate ω and δ for a separated passage, regardless of i and i_S . These new correlations are based on the theory by Thomason and Moses [62] for separated passages. Although these correlations model a separated passage, their use in the code is analogue to that of normal forward flow correlations. They are not the same as the correlations for reverse flow, described in the next section.

The criteria was developed for steady-state and is extended to transient simulation using a time-lag law. A virtual passage is modelled as stalled only if the right inlet and outlet conditions occur for a sufficiently long time for the stall to develop. This is practically obtained by the fact that ω and δ will reach the new higher value in the time set by the time constant τ .

4.7 Separated flow modelling

The modelling of separated (reverse and mixed) flow is crucial for the post-stall simulation. It must be accurate but also robust enough to avoid numerical instability during the chaotic reversing of the flow. Models of separated flow have been developed by Moses and Thomason [62] and recently by Longley [41], and they are all based on similar assumptions:

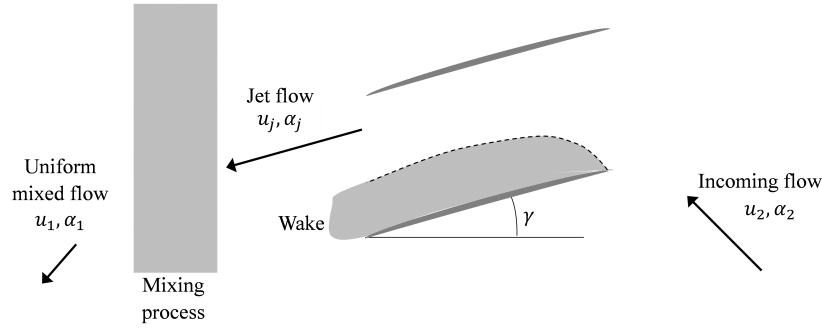


Fig. 4.20: Separated flow structure within a typical blade passage in reverse flow, based on the model proposed by Longley [41].

- Flow separates on the suction surface and the mixing can continue after the trailing edge,
- The separated jet exits at the T.E. (or L.E. in reverse) with the direction of the stagger angle,
- Friction is negligible and the jet flow is isentropic, pressure loss derives only from separation and mixing,
- Deviation is due to the wake mixing.

In the model proposed by Longley the separation, and therefore the pressure loss, is produced by the force which abruptly turns the flow when it enters the blade passage at high angle. This force is the integral of the pressure on the blade surface and is always normal to the blade. As the turning force is directed against the incoming flow it also introduces a pressure loss. The flow does not slow down, but separates isentropically and the loss is not detectable until mixing occurs.

A similar approach is used in the current code: when the mixed or reversed flow is present the normal and parallel forces are replaced by only one turning force which is normal to the blade, not to the flow angle. The change in direction is mainly provided by the correction force F_{COR} . ω is not calculated, but the pressure loss is introduced *naturally* by the turning force.

This method is preferred to the use of empirical correlations as it allows to deal with mixed flow when an inlet and an outlet are not clearly defined and strong radial flow is present.

The calculation of deviation has been developed starting from Longley's model described in the literature, section 3.2.3. The parameter b is defined in his model as the ratio between the real axial momentum of the separated flow and the axial momentum of the pitchwise-averaged flow. It can be calculated from the inlet attack angle as:

$$b = 1 + \left| \frac{\sin(\alpha_2 - \gamma)}{\cos(\alpha_2)} \right| \quad (4.52)$$

By applying a control volume to the flow between states j and 1, (figure 4.20), and by solving the momentum in the axial and tangential direction, the flow angle α_1 can be calculated as

$$\tan(\alpha_1) = \tan(\alpha_j) \cdot b \quad (4.53)$$

From the angle at the exit the deviation can be calculated. Note that the flow is considered incompressible and thus the density is assumed constant in the passage.

The resulting model of the separated flow then becomes the one shown in figure 4.21. The pressure loss takes place only at the entrance, where the flow is misaligned, and a deviation is in place to impose the estimated post-mixing angle.

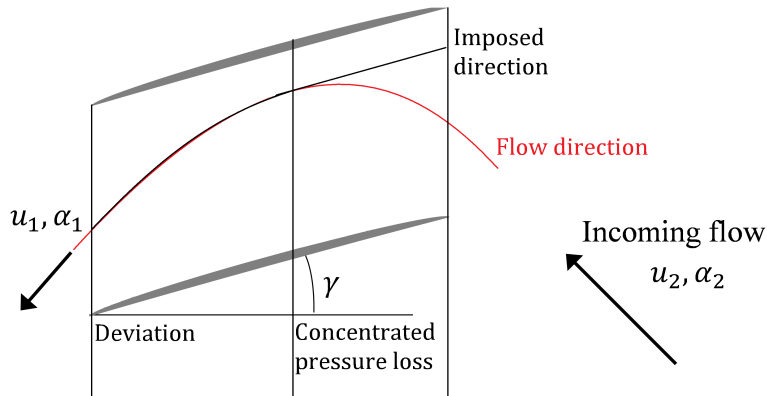


Fig. 4.21:]
Scheme of the novel separated flow passage treatment.

While the model in figure 4.20 considers the flow in the passage to be fully isentropic in reality the mixing occurs partly inside of the blade at the interface between the separated boundary layer and the jet. This is considered in Longley's model by

means of an *entropy-generation* term which, in every element where separation is present, mixes out the flow at a prescribed ratio. The prescribed ratio is such as the *excess energy*, which is the energy stored in the non-uniformity of the flow, is mixed in the time in which the jet flow travels the distance of a half a pitch.

The mixing out occurring inside of the blade passage is introduced in the correlation used for estimating the deviation. Therefore the deviation becomes function also of the length and pitch of the passage:

$$\delta = f(\alpha_{IN}, \gamma, \sigma) \quad (4.54)$$

The detailed description of the formulation of the new correlation and its derivation are reported in appendix B.

The assumption of incompressibility sets a limit to the model. As the Mach number increases we expect that a large difference in flow velocity between inlet and outlet can occur, depending on the angles. With a different flow velocity at the outlet the value of b at the trailing edge would change and so would the post-mixing deviation. An extension of the method to compressible flow would necessarily require a further development of this model.

4.8 Boundary conditions

Different types of boundary conditions are used to simulate the compressor domain, figure 4.22. A plenum and throttle model can also be added to the simulation as it allows to simulate dynamically changing conditions downstream of the compressor. In this section the details and formulation of each of them are described.

4.8.1 Non-reflective boundary conditions

Very specific boundary conditions are necessary at domain inlet and outlet in order to model the post-stall transient behaviour of the compressor. They must be able to work during flow reversal, with subsonic and supersonic regimes and they must behave as physically as possible. Different types of conditions for boundaries with

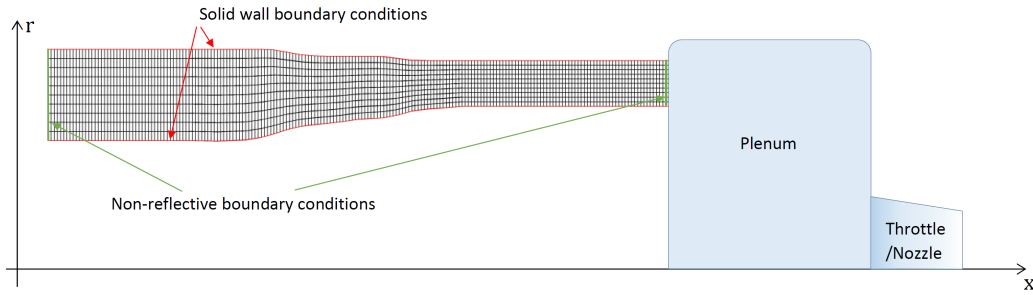


Fig. 4.22: Boundary conditions applied to the compressor's computational domain.

mass exchange have been considered. These are presented for comparison in figure 4.23.

The most common type of boundary conditions are obtained by imposing the prescribed value of pressure and temperature as constant at the boundaries. During transient simulations these so called *reflective* boundaries reflect every wave exiting the domain generating new non-physical waves travelling backward.

Another simple type of boundary conditions can be implemented by using ghost nodes outside the boundary and extrapolating every flow property from inside the domain. When these boundary conditions are crossed by a wave they will let it pass through and adapt to the flow conditions which are behind the wave. In this case no reflection is generated. The flow properties at the boundary however can not be imposed any more but they depend only on what happens inside the domain.

The boundary conditions used in this code are the *non-reflective boundary conditions* proposed by Poinso and Kele [63]. These boundaries adapt to the flow property behind the exiting wave like those obtained by extrapolation but then slowly return to the imposed value.

The non reflective boundary conditions are imposed by modifying the Euler equations solved in all the elements at the first and last axial positions of the domain. For example at the last axial position in the domain the equations used are:

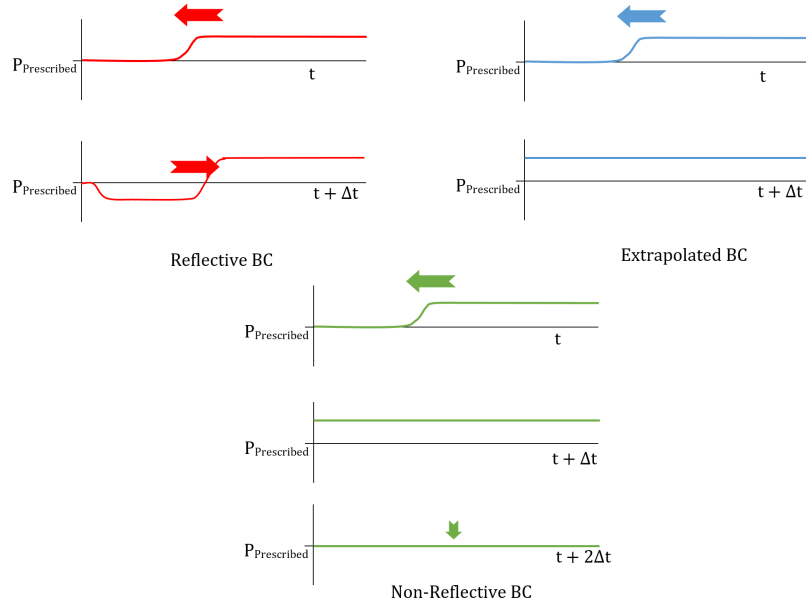


Fig. 4.23: Types of boundary conditions and their dynamic response to a pressure wave exiting the domain.

$$\begin{aligned}
 \frac{\partial}{\partial t} \begin{bmatrix} \rho \\ \rho u_r \\ \rho u_\theta \\ \rho u_x \\ \rho E_0 \end{bmatrix} &= -\frac{1}{r} \frac{\partial}{\partial r} \begin{bmatrix} \rho r u_r \\ \rho r u_r^2 \\ \rho r u_\theta u_r \\ \rho r u_x u_r \\ \rho r H_0 u_r \end{bmatrix} \cdot I_{SW} - \frac{1}{r} \frac{\partial}{\partial \theta} \begin{bmatrix} \rho u_\theta \\ \rho u_r u_\theta \\ \rho u_\theta^2 + P \\ \rho u_x u_\theta \\ \rho H_0 u_\theta \end{bmatrix} \cdot I_{SW} + \begin{bmatrix} d_1 \\ u_r d_1 + \rho d_5 \\ u_\theta d_1 + \rho d_4 \\ u_x d_1 + \rho d_3 \\ H' \end{bmatrix} + \begin{bmatrix} 0 \\ \frac{\rho u_\theta^2}{r} + \frac{\partial P}{\partial r} \\ -\frac{\rho u_\theta u_r}{r} \\ 0 \\ 0 \end{bmatrix} \\
 &\quad (4.55)
 \end{aligned}$$

where:

$$H' = 0.5(u_r^2 + u_x^2 + u_\theta^2)d_1 + d_2/(\gamma - 1) + \rho u_x d_3 + \rho u_\theta d_4 + \rho u_r d_5$$

At the elements in the first axial position a similar set of equations is used although with different signs as the direction normal to the boundary is negative. The vector in red is used to introduce the boundary condition. It replace the flux in the axial direction and is calculated with a finite difference method. The flux terms

in the circumferential and radial directions and the source terms are solved using the same finite volume method as the rest of the domain. By construction the grid at the domain boundary is perpendicular in the meridional plane therefore there is no interference between the axial flux and the others.

The switch I_{SW} is used to control radial equilibrium. The switch is set to 1 only where the flow is exiting the domain. In this case the presence of the radial flux in these equations guarantees that the radial equilibrium of pressure is maintained and, in case of high swirl, the pressure has a profile in the span-wise direction. With the radial equilibrium active, only the average static pressure will equal the one imposed. The switch is set to 0 where the flow is entering the domain. In this case each point along the span will reach the prescribed value of inlet total pressure and temperature at that position. The same method is used with the circumferential flux in case of asymmetric flow. The switch is calculated locally ($I_{SW} = I_{SW,j,k}$) and can deal with mixed flow where part is exiting and part is entering the domain.

The terms d_1, d_2, d_3, d_4, d_5 are the incoming and outgoing characteristic wave amplitudes. The formulation of this wave amplitude contains the imposition of the prescribed values of pressure, temperature and direction. More details on their formulation are given in appendix C.

The number of flow property imposed at the boundary depends on the local flow:

- For flows entering the domain the total pressure and total temperature are imposed. The direction is imposed as axial.
- For flows exiting the domain subsonically only the static pressure is imposed.
- For flows exiting the domain at supersonic speed nothing is imposed, and the boundary condition behaves similar to the extrapolated type (see figure 4.23).

If the flow is exiting supersonically but the static pressure at the boundaries becomes lower than the prescribed value, the algorithm switches back to subsonic boundary conditions. This forces the flow to decelerate and re-pressurise through a shock-wave travelling backward.

4.8.2 Wall boundary conditions

Wall boundary conditions are applied at every element on the first and last radial position. Unlike the boundary conditions at inlet and outlet they are not applied by modifying the equations but by creating ghost nodes. The ghost nodes (in this case more correctly ghost volumes) are elements added on the other side of the solid wall which only serve to impose the boundary condition.

In case of the hub wall the elements in contact with it have a radial index $j = 1$; ghost nodes are created on the other side of wall with index $j = 0$.

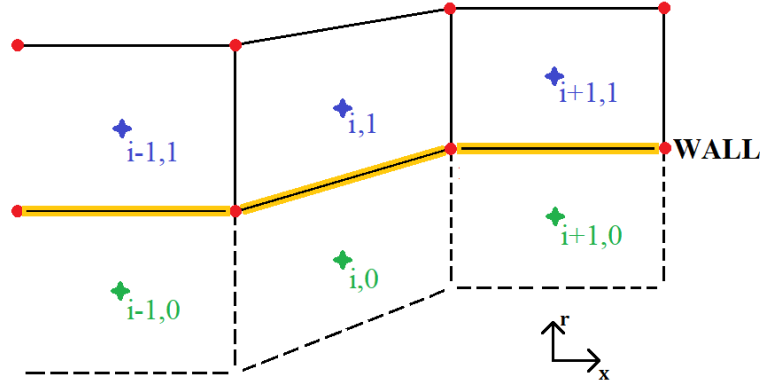


Fig. 4.24: Ghost nodes at the hub wall (in green).

The values of pressure, density and temperature in the ghost node are set to be the same as their neighbour:

$$P_{i,0,k} = P_{i,1,k} \quad T_{i,0,k} = T_{i,1,k} \quad \rho_{i,0,k} = \rho_{i,1,k} \quad (4.56)$$

The component of velocity normal to the solid wall (which coincides with the inter-element boundary) is set as the opposite of the value inside the domain, while the tangential and the circumferential are set to be the same:

$$u_{\perp i,0,k} = -u_{\perp i,1,k} \quad u_{\parallel i,0,k} = u_{\parallel i,1,k} \quad u_{\theta i,0,k} = u_{\theta i,1,k} \quad (4.57)$$

The ghost nodes are used when solving the Riemann problem at the solid wall boundary. From the result the primitive variables at the boundary $V_{i,0,k}^{RS}$ are ex-

tracted and used in the Euler equations.

The tangential velocity at the wall is the same as in the element centroid and in fact a free-slip condition is introduced. This is the only possibility since no shear stress is modelled by the Euler equation and no-slip conditions would be impossible.

4.8.3 Plenum and throttle model

As was first introduced by Greitzer [33] a plenum and throttle model can be applied downstream of the compressor domain. This model does not substitute the boundary conditions described in the previous section 4.8.1, but it allows to modify dynamically the value of pressure and temperature prescribed making it dependent on the mass-flow. This type of boundary is necessary to model surge and rotating stall as they happen in a compressor rig where, in fact, a plenum and throttle are physically present. The effect of the plenum on compressor stall has been described in chapter 3.

Following the model of Greitzer, the plenum is considered to be much smaller than the wavelength of an acoustic wave associated with surge. Therefore the static pressure in the plenum is uniform and the velocities are negligible. The variation of density and pressure inside the volume depends on the mass-flow exchanged with the compressor $\dot{m}_C$ and with the atmosphere through the throttle $\dot{m}_T$. The density variation is calculated as

$$\frac{\partial \rho_{Pl}}{\partial t} = \frac{1}{V_{Pl}}(\dot{m}_C - \dot{m}_T) \quad (4.58)$$

Assuming that the pressure in the plenum changes with an adiabatic transformation, the pressure variation can be written as

$$\frac{\partial P_{Pl}}{\partial t} = \frac{\gamma P_{Pl}}{\rho_{Pl} V_{Pl}}(\dot{m}_C - \dot{m}_T) \quad (4.59)$$

The value of $\dot{m}_C$ is taken as the instantaneous integral of mass-flow at the computational domain outlet. The mass-flow through the throttle is calculated considering it as a simple convergent nozzle discharging from the plenum pressure to the atmospheric pressure P_{ATM} . The atmospheric pressure is input from the user and is not

affected in any way from the flow in the simulation. The Mach number at the nozzle throat is calculated as:

$$M_T^2 = \frac{2}{\gamma - 1} \left(\frac{P_{Pl}}{P_{ATM}}^{\frac{\gamma-1}{\gamma}} - 1 \right). \quad (4.60)$$

The value of M_T is limited to a maximum value of one as the nozzle would choke.

The mass-flow in the throat is

$$\dot{m}_T = \sqrt{\gamma \rho_{Pl} P_{Pl}} \cdot A_T \cdot M_T \left(1 + \frac{\gamma - 1}{2} M_T^2 \right)^{-\frac{\gamma+1}{2(\gamma-1)}}. \quad (4.61)$$

At every time step a 4th order Runge Kutta method is employed to calculate the time evolution of ρ_{Pl} and P_{Pl} . The scheme used to advance from time t to $t + dt$ is

$$U_{Pl,0} = \begin{bmatrix} \rho_{Pl}^t \\ P_{Pl}^t \end{bmatrix} \quad (4.62)$$

$$\begin{aligned} \left(\frac{\partial U_{Pl}}{\partial t} \right)_1 &= f(U_{Pl,0}) \\ U_{Pl,1} &= U_{Pl,0} + \frac{1}{2} \left(\frac{\partial U_{Pl}}{\partial t} \right)_1 \cdot dt \end{aligned}$$

$$\begin{aligned} \left(\frac{\partial U_{Pl}}{\partial t} \right)_2 &= f(U_{Pl,1}) \\ U_{Pl,2} &= U_{Pl,0} + \frac{1}{2} \left(\frac{\partial U_{Pl}}{\partial t} \right)_2 \cdot dt \end{aligned}$$

$$\begin{aligned} \left(\frac{\partial U_{Pl}}{\partial t} \right)_3 &= f(U_{Pl,2}) \\ U_{Pl,3} &= U_{Pl,0} + \left(\frac{\partial U_{Pl}}{\partial t} \right)_3 \cdot dt \end{aligned}$$

$$\left(\frac{\partial U_{Pl}}{\partial t}\right)_4 = f(U_{Pl,3})$$

$$U_{Pl}^{t+dt} = U_{Pl,0} + \left(\frac{1}{6}\left(\frac{\partial U_{Pl}}{\partial t}\right)_1 + \frac{1}{3}\left(\frac{\partial U_{Pl}}{\partial t}\right)_2 + \frac{1}{3}\left(\frac{\partial U_{Pl}}{\partial t}\right)_3 + \frac{1}{6}\left(\frac{\partial U_{Pl}}{\partial t}\right)_4\right) \cdot dt$$

Where the time derivative is calculated using equations (4.58 - 4.61). $\dot{m}_C$ is estimated once at the beginning, $\dot{m}_T$ is estimated at every intermediate time step. At every time step (of the main Euler solver) P_{Pl} is used as the imposed static pressure at the compressor outlet.

In case of flow reversal, P_{Pl} is the prescribed total pressure while the temperature $T_{Pl} = P_{Pl}/(\rho_{Pl}R)$ is the total temperature. If during a surge the plenum pressure P_{Pl} equals the atmospheric one P_{ATM} and the mass-flow through the nozzle becomes 0, then the algorithm will impose the pressure and temperature in the plenum to be the same as the atmospheric. This practically means that temperature and pressure at the compressor outlet equals the atmospheric, without any losses through the throttle.

When the plenum and throttle model is used, the main control of the simulation is the throttle area A_T which can be dynamically changed to simulate the typical operations of a compressor rig. The atmospheric pressure at the nozzle exit and the inlet conditions can also be changed.

The plenum and throttle can be used to simulate the combustion chamber and turbine NGV of a real engine. However, the lack of a model for combustion does not allow to model the post-stall behaviour of a real engine with all the possible outcomes, as was done by Wilson [42]. The modelling of combustion would also permit to simulate the fuel spike as a trigger for compressor stall; in the present code the variable throttle is used instead.

4.9 Simulation input, output and strategy

This section is devoted to explain how the simulation is set-up and controlled by the user. The first part explains which inputs are required and what type of simulation

can be chosen. The second part describes how the outputs are created and how they are post-processed. In the last part is described the strategy used to obtain rotating stall and surge. A flow chart describing the order of the code algorithms is reported in appendix D.

4.9.1 *Simulation input*

The input consists of a series of .txt files which are read sequentially by the program. The first file specifies the general settings:

- The simulation mode: *transient* or *steady*.
- The axial, radial and circumferential size of the domain.
- The total length in time of the simulation and the intervals at which the flow-field is saved. These are both input in μs .
- The use of a plenum/throttle model or a constant outlet BC at the outlet. NB the plenum/throttle model can be used **only** if the simulation mode is *transient*.
- The correlations set used for forward flow, choosing between those described by Aungier or those developed by Rolls-Royce.
- The number of cores employed for the simulation.

The *steady* simulations start from a uniform flow-field in all the domain. In this mode an algorithm checks at every time step the axial mass-flow across the domain; when the mass-flow is uniform (under a certain threshold) the simulation is considered converged. The converged simulation is stopped and the flow-field saved.

The *transient* simulations load the flow-field from a saved result and use it as a starting point. This can be a previously converged solution or any saved result, even in the middle of a transient. If this mode is chosen the data to be loaded must be present in the same folder as the input, this includes the 5 primitive variables' values in all the domain, the location of the grid element centres and the number of elements in the 3 directions of the simulation loaded. All these data files are

produced by the tool output in the correct format and they just need to be copied and pasted.

The compressor geometry is set up by three input files, they specify:

1. The annulus shape, input as the $x - r$ coordinates of the points forming the hub and tip wall.
2. The position and shape of the blade rows in the meridional plane. For every blade row the type (rotor or stator) and the number of blades are also specified.
3. For every blade row a series of blade profiles from hub to tip. For each profile the inlet and outlet metal angle, the stagger angle and the type of profile are input.

The types of blade profile recognised are *double circular arc* and *parabolic arc*. From the type and the input angles the camberline of the whole profile is generated. In every blade row then we have many profiles at different span positions. From these the local value of the camberline in every element centroid is interpolated. These values form the β_0 profiles used to generate the imposed direction at every j pseudo-plane as described in section 4.3.3.

These three geometric input files have the same format of those used by the program SOCRATES. Therefore any geometry created for one code can be easily used in the other.

Another input file is dedicated to the boundary conditions. In this file the initial value of total temperature, total pressure at inlet and static pressure at outlet are specified. These are input as radial profiles from hub to tip. The use of profiles allows to reproduce specific inlet conditions from experiments or CFD for better comparison. As described in section 4.8 these profiles are ignored in the boundaries where the flow is exiting the domain and the radial equilibrium is imposed. The initial value of rotational speed in RPM is also specified.

After the initial conditions a list of *events* can be added. Each one of these events prescribes a change in boundary conditions at a specific time t . Both rotational speed and pressure/temperature profiles can be modified. While the initial conditions are

set identical at every circumferential position, the subsequent modifications can be applied to circumferential intervals, creating non-symmetrical perturbations.

If a plenum and throttle model is used then an additional input file is required. In this input file the atmospheric pressure, the plenum volume and the throttle area are specified. When using a plenum, the value of static pressure at the outlet prescribed in the previous input file is discarded. If the throttle area is specified as -1 then it is calculated from the value of mass-flow and pressure at the initial instant.

$$M_T^2 = \frac{2}{\gamma - 1} \left(\frac{P_{Pl}}{P_{ATM}}^{\frac{\gamma-1}{\gamma}} - 1 \right) \quad (4.63)$$

$$A_T = \frac{\dot{m}_C}{\sqrt{\gamma \rho_{Pl} P_{Pl}} M_T} \cdot \left(1 + \frac{\gamma - 1}{2} M_T^2 \right)^{\frac{\gamma+1}{2(\gamma-1)}} \quad (4.64)$$

Also in this input file a list of events can be added, each one prescribing a change in atmospheric pressure or a change in throttle area. The changes in atmospheric pressure require the new value of pressure, the changes in throttle area instead are input as percentages $\Delta A_{\%}$ meaning that in the event at time t :

$$A^{t+dt} = A^t \cdot \Delta A_{\%} \cdot 100. \quad (4.65)$$

The automatic throttle creation should be used only if the flow-field is in steady state. If a unsteady transient simulation is loaded (for example a developed rotating stall) it is necessary to load also the conditions in the plenum and the throttle area. For this reason whenever the simulation is saved also the values of ρ_{Pl} , P_{Pl} , V_{Pl} , A_T are saved in a separate file called 'TANKDATA'. If when the simulation is restarted this file is present, then it is loaded and these plenum conditions are used. Otherwise the normal procedure is followed and, if requested, a new throttle area is calculated. Even if the plenum conditions are loaded from the results (and the data in the plenum input file discarded) the events list is still read, to allow for further throttle or atmospheric pressure changes.

4.9.2 Simulation output

The main output of the code is the flow-field itself, consisting of the values of the primitive variables V in every element in the domain. These are saved at the prescribed time interval in successive folders each containing the whole flow-field at that time step. The output results are printed in extension-less files written in ASCII code (i.e. the numbers are written in characters). Aside from the 5 primitive variables also the grid and the blade geometry are saved, but this is done only once at simulation start.

Other data can be saved as required by the user, for example ω and δ , the flow regimes, the direction β in every element and many other parameters. The size of the output data can be a limiting factor when deciding which parameters to save. A 3D simulation with 512000 elements modelling 800 ms of surge with results printed every 2 ms requires 61 GB of space with all parameters saved. The same simulation would take about 20 GB with only the primitive variables saved.

The text-like results are post-processed using MATLABTM scripts. These include:

- performance analysis used to create maps and draw operating point trajectories,
- visualization of the flow-field along planes in the 3 directions or with probes mimicking the experimental measurements.

The scripts can be used to analyse a particular moment in time or they can be programmed to run through every folder and draw the evolution in time of the flow-field. A library of MATLAB scripts has been developed by the author and can be used to quickly obtain some standard post-processing.

4.9.3 Simulation strategy

This section contains the strategy developed to obtain a full compressor map for a new compressor geometry in the most computationally efficient way possible.

When a new compressor geometry is studied the first step is to obtain forward and reverse characteristics. The simulations are launched from uniform conditions (*steady mode*) and the rotational speed and boundary conditions are set constant. Every simulation is run until a steady state is obtained; and each one of the points is then used to form the characteristic. It is normally a good habit to use the flow-field post-processing to check the result obtained is as expected. Since in these two regimes the real flow-field is mostly axi-symmetric the simulations are run in 2D to speed them up. This is obtained by setting the circumferential elements N_θ to 1. The solver will then ignore every circumferential flux and rotational term running a faster, symmetric version of the code.

To quickly create the characteristics, the simulation can be programmed with many subsequent events. For example when reverse flow is simulated the initial gradient of pressure is set very high to guarantee that the compressor will stall and enter reverse flow. After the compressor reaches a steady operation on the characteristic, the programmed events will gradually reduce the gradient of pressure between boundaries. If the intervals are large enough that the simulation reaches a steady state between events, then with a single run the full characteristic can be created.

When an high-speed compressor is modelled the first simulation can require longer computational times. If the compressor is modelled with a high starting speed ($>\sim 4000$ RPM) and a uniform flow-field then the pressure waves caused by the initial adjustments will likely crash the code. If the code is robust enough not to crash, there is also a chance that the compressor will stall because of the large pressure waves travelling in the domain. If the objective was to obtain the compressor running in forward flow then the simulation would need to be restarted. The same problem occurs if a large change in rotational speed is requested as a single event. The changes are imposed instantaneously, and this can cause an excessive and unstable reaction in the flow-field. The strategy normally used is starting the simulation at low speed (around 3000 RPM) and use subsequent steps to gradually increase it to full speed.

In the case of the NASA compressor rig described in section 6.3 the 70% speed-

line, just below transonic conditions, can be reached with subsequent steps in only 100 ms (simulated time). While it might look like a slow process, an acceleration of this magnitude obtained in only 0.1 s is a very un-physical and extreme situation and requires a very robust code to be simulated. With a 2D simulation this acceleration can be performed in 3 minutes of computation on a quad-core workstation. The up-to-speed solution can then be saved and reloaded as starting point for the next simulations at that speed.

To simulate a compressor stall the starting point is usually a converged simulation of an unstalled operating point close to the surge margin, obtained in 2D. The 3D stall simulation is run in *transient* mode and loads the flow-field from the 2D solution. The 2D flow-field is copied at every θ position and creates a symmetrical initial condition. To simulate a compressor stall a plenum and throttle model **must** be used. To initiate the stall a boundary event is used; this can be either a throttle closure, a change in inlet condition, a change in atmospheric conditions (at nozzle exit), or a change in rotational speed. Depending on the conditions imposed, the various type of compressor stall can be obtained.

The 3D simulations, which start perfectly axi-symmetric, never develop any asymmetry since all the circumferential positions in the numerical model are exactly the same. To introduce some asymmetry a temporary pressure distortion is set on part of the inlet using boundary condition events. Even if the distortion lasts only few milliseconds this was found to be enough, in the right conditions, to develop a rotating stall.

Rotating stall takes only few rotor revolutions to develop but, if a large number of elements is used, this can require more than 15 hours of calculation. To speed up the creation of the rotating stall characteristic only the first point is obtained as a result of a compressor stalling. All the subsequent simulations are started loading the flow-field of the previous rotating stall. By closing and opening the throttle the other points on the characteristic are obtained in a shorter time (figure 4.25).

Once enough rotating stall operating conditions are simulated a characteristic can be drawn, and the full compressor map is completed. Figure 4.26 is a diagram summarising the steps used to test a new compressor.

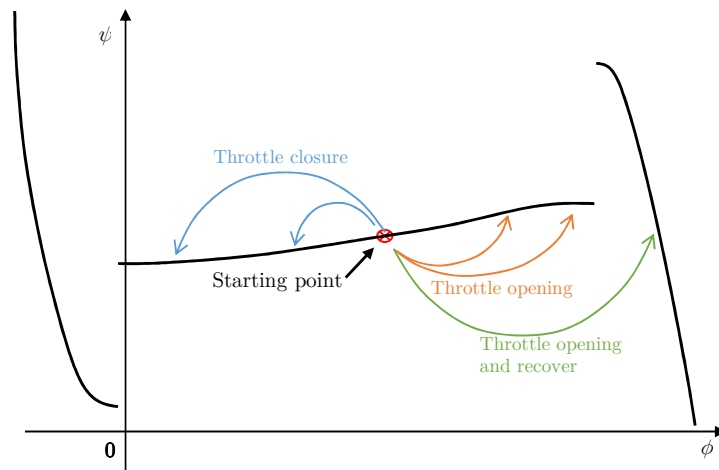


Fig. 4.25: Trajectory of rotating stall operating point with throttle opening and closure.

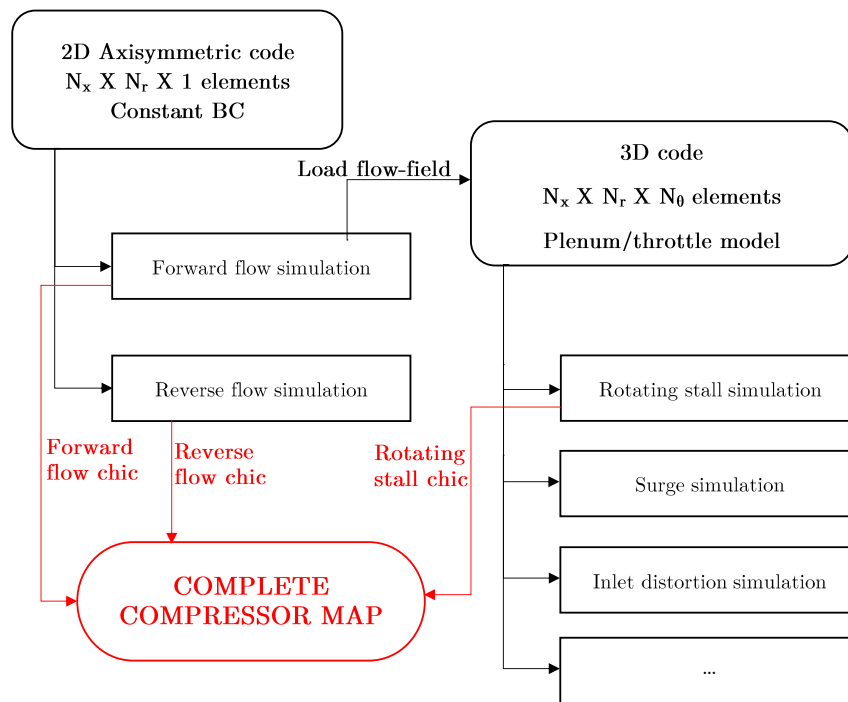


Fig. 4.26: Strategy used to test the post-stall behaviour of a new compressor design.

4.10 Programming

The code is written in Fortran90 and programmed using *Microsoft Visual Studio* as IDE (Integrated Development Environment) and *Intel Parallel Studio XE* as compiler. The empirical correlations for forward flow imported from SOCRATES

are pre-compiled DLLs (Dynamic Link Libraries) and are kept as such in the present code. The new correlations produced during the project are not DLLs but integral part of the code; as such they are easy to access, modify and test. Compiling of the correlations into DLLs will be performed once their development is concluded.

The program has been parallelised using OpenMP directives in order to reduce the computational cost. Because of its architecture, the Euler solver is easy to convert into parallel computing. The domain is divided in the axial direction in sub-domains which are processed by separate processors. Each processor runs almost independently its own sub-domain. Each processor calculates the Riemann solvers, local B.C., fluxes, time derivatives and flow-field at new time-step $\mathbf{V}^{t+dt}$ in its own domain without needing data from the others.

An exception is represented by the fluxes at the interface between the sub-domains. Each of them is calculated by a particular processor but used also by the processor working on the adjacent sub-domain. If a processor lags behind and has not finished calculating the fluxes, the adjacent processor might use a faulty value in his own calculation. To avoid this problem a *barrier* is introduced after the flux calculation. This barrier will hold the processors in that point until they have all reached it. Once all the processors have cleared the fluxes section, they will start with the second part of the solver.

Although the Euler solver is only one of the algorithms, it consumes a large part of the computational time of every step, $\sim 80\%$. At this stage, it was not considered necessary to extend the parallelization to the body force method or the correlations.

The computational time obtained with different cores is shown in figure 4.27, the results were obtained on a work station with a quad-core processor and 8 virtual threads. If more than 4 threads are used, then 2 of the threads are run by the same core at half speed. Therefore there is no substantial improvement in computational time when using more than 4 cores.

Although a large acceleration is obtained, the benefit rapidly decreases with the number of processors, with very little differences between 3 and 4 processors. As such, the current parallelisation will probably not be efficient on a large number of processors (like a cluster). If the code is to be run on more processor than

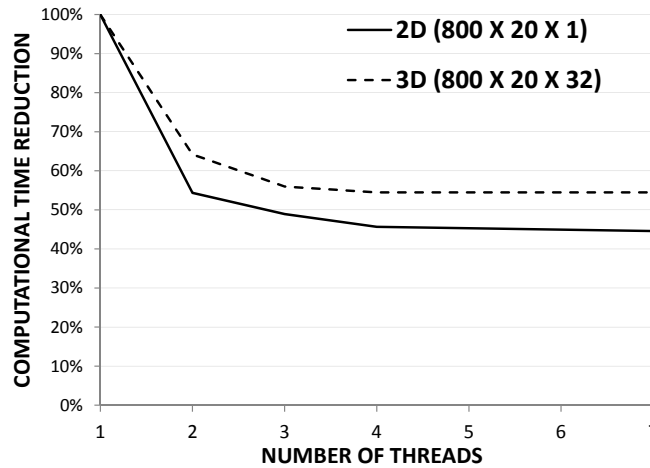


Fig. 4.27: Computational time reduction with number of threads employed.

a workstation, a better parallelisation should be implemented. For example the parallelization should be extended to other algorithms, like the body force method. In fact, as the Euler solver is parallelised on many cores the body force method becomes the next most computational ‘costly’ part of the simulation.

4.11 Summary

The literature review presented in this thesis has identified a specific set of requirements for the new tool developed in this research. The type of tool created was chosen to be a three-dimensional through-flow code using the body force method and empirical correlations; particular attention was devoted to the method used to model the performance of separate blade rows during reverse flow. This chapter presented the development of the new through-flow tool and its formulation. The various algorithms implemented in the new tool were described in detail, highlighting their assumptions and limitations.

Some of the methodology here reported represents a contribution to knowledge.

- The Euler solver selected for the tool, using the Godunov scheme, is employed here for the first time in the modelling of three-dimensional compressor stall in a through-flow code. The advantages of using this type of scheme in the current application have been presented and discussed.

- A new method for the modelling of separated blade rows during reverse flow operation has been proposed. This method is based on the model for separated blade rows proposed by Longley [41], but estimates the performance of the blade rows using empirical correlations and a modification of the body force method.

In the following chapter the code created will be validated using various test cases for forward flow, reverse flow and compressor stall, following the steps proposed in chapter 2.

The reduction of the computational cost was another important focus of the tool development; in fact, the tool has been parallelised and a simulation strategy has been proposed to create the full compressor map with the minimum computational cost. During the validation, the full compressor map of two multi-stage low-speed compressor rigs will be created, assessing the computational cost performing this task using the new tool.

Chapter 5

Test case geometries and set-up

In this section the set-up of the test cases and their corresponding models are described. Special focus is given to which informations were or were not available and which assumptions have been made. In this thesis 4 test cases are analysed, they will be called:

- 2-stage NASA rig
- 3-stage MIT rig
- 4-stage C106 rig
- Modern high-speed HP compressor

5.1 2-stage NASA rig

The *2-stage NASA rig* is a transonic experimental compressor tested for performance by Urasek et al. [61].

The model was set up easily as the geometry of the compressor is described in detail and every information needed is available. The compressor has a low hub-to-tip ratio and an increasing mean-line annulus. The modelled annulus is extended at constant span of about two radii in front and at the back of the compressor.

The compressor performance is also described in detail; in particular pressure ratio, PR , and mass-flow, $\dot{m}$, are available on several points along the 50%, 70%, 80%, 90% and 100% speed-lines. In the post-processing of the tool simulations, the pressure ratio is taken as the ratio of the area-averaged total pressure at domain

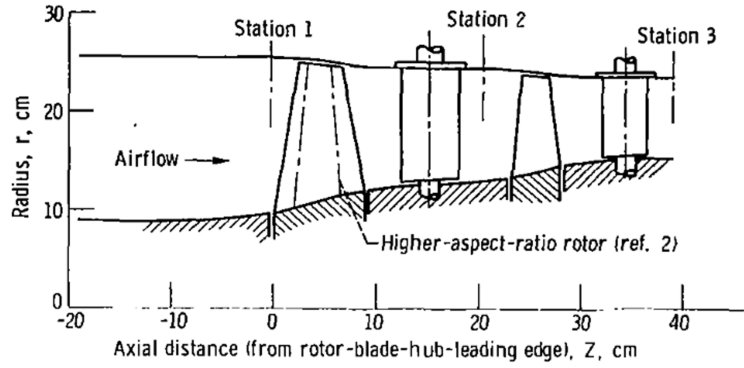


Fig. 5.1: Meridional view of the NASA 2-stage rig [61].

outlet ($i = N_X$) and domain inlet ($i = 1$)

$$PR = P_{0\text{OUTLET}}/P_{0\text{INLET}} \quad (5.1)$$

The mass-flow is calculated at the domain inlet, as the sum of the mass-flow in all elements with $i = 1$

$$\dot{m} = \sum_{j=1}^{N_r} \sum_{k=1}^{N_\theta} u_{x\ 1,j,k} \cdot \rho_{1,j,k} \cdot A_{1,j} \quad (5.2)$$

5.2 3-stage MIT rig

The *3-stage MIT rig* is the low-speed compressor described in section 3.3.1. Gamache [57] tested two different geometries obtained by re-staggering the blades. In the current work only the low reaction build was modelled, the same geometry also used by Eastland [56]. Only little geometrical information were given by Gamache and Eastland. The blade angles are reported only at mid-span but from the pictures in the report (figure 5.2) it appears that the blades are slightly twisted. The only information on the blades' shape in the meridional plane is the mean chord. Therefore the blades in the model are assumed to be square and untwisted. The meridional position of the blades is taken from the paper of Haynes et al. [64] who also tested this rig for active rotating stall control.

The compressor has a constant hub and tip annulus. The modelled domain is extended at constant hub and tip of one hub radius in the front and back of the

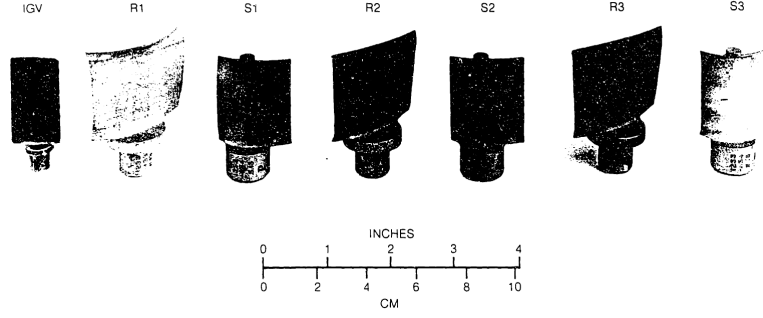


Fig. 5.2: Blades from the 3-stage MIT rig [57].

blade rows.

The performance maps are reported as pressure rise ψ over flow coefficient ϕ . Data are available with static-to-static, total-to-static and total-to-total pressure rise coefficient; in the present work only the total-to-total ψ_{T-T} will be used, unless otherwise specified. The pressure rise coefficient is defined as:

$$\psi_{T-T} = \frac{\Delta P_0}{0.5\rho\bar{U}^2} \quad (5.3)$$

where the density is measured as the area-average at compressor inlet, the blade speed $\bar{U}$ is taken at mean radius and the delta pressure is measured from domain inlet to outlet. The P_0 and T_0 are area-averaged at the domain inlet and outlet. The flow coefficient is defined as:

$$\phi = u_x/\bar{U} \quad (5.4)$$

where the axial velocity used the area-averaged value at domain inlet.

The torque measurements during steady reverse flow and rotating stall are also reported using a torque coefficient defined as:

$$\psi_{\mathcal{T}} = \frac{\mathcal{T}}{\frac{1}{2} \cdot \rho U^2 A_C \bar{r}} \quad (5.5)$$

where $\mathcal{T}$ is the integral of the torque on all the rotors.

5.3 4-stage C106 rig

The C106 rig is a low-speed compressor with interchangeable blades, property of the Whittle laboratory in Cambridge University. The blade set-up modelled is the one tested by I. Day in 1994 [58] and described in section 3.3.2. The blade angles reported in the paper are only the camber angle and the stagger which do not describe the geometry (or at least the turning) uniquely. The full design tables of the blades have been kindly supplied privately by Professor Day to this author; this geometry is propriety of Rolls-Royce and will not be described in this thesis. The data supplied are for one stage, repeated 4 times; each blade row is described by 5 profiles from hub to tip. The IGV geometry however was not described neither in literature nor privately. The IGV was then assumed to have 0° inlet angle and the same T.E. angles as the stators, which makes them very lightly loaded. The IGV stagger angle was guessed as half of the T.E. angle.

The annulus has constant hub and tip, the domain was extended with straight walls of 2 radii in front and 1.5 radii behind the compressor.

The compressor maps are also provided as $\phi - \psi$ but the post-processing is different from the one employed by Gamache and Eastland. In fact the pressure jump is calculated as the difference between the total pressure at inlet and the pressure (static) in the plenum, and the denominator does not have the coefficient $1/2$. The pressure rise coefficient is then

$$\psi = \psi_{T-S\ C106} = \frac{P_{Pl} - P_{0\ INLET}}{\rho \bar{U}^2} . \quad (5.6)$$

5.4 Modern high-speed HP compressor

A modern high-speed 6-stage HP compressor is modelled to investigate the applicability of the current code to commercial turbo-machinery rather than experimental rigs. The geometry was available in detail to the author and no assumption was necessary. This data is proprietary information of Rolls-Royce plc and will not be reported or represented in this thesis.

Only the characteristics at reverse flow were modelled, these were available as pressure and temperature ratios over corrected mass-flow. The corrected mass-flow during reverse flow is defined as:

$$\dot{m}' = \dot{m} \cdot \frac{\sqrt{T_{0OUTLET}}}{P_{0OUTLET}} \quad (5.7)$$

where the temperature and pressure are taken at the domain (geometric) outlet.

The compressor data are Rolls-Royce property and commercially sensitive, therefore the results have been scaled by factors K_{SCALEX} and K_{SCALEY} . An offset $K_{OFFSETY}$ has been added and the rotational speed has been non-dimensionalised with a reference NRT^* .

5.5 Non-dimensionalization of characteristics and transients

A non obvious choice had to be made in the definition of u_x and ρ used in the calculation of ϕ and ψ for the 3-stage MIT and 4-stage C106 cases.

When testing the 3-stage MIT rig both Eastland and Gamache used an orifice plate to measure the mass-flow, and both of them calibrated the flow using flow measurement at the facility's inlet (see section 3.3.1). However since Gamache physically inverted the compressor, in his case the ϕ is based on the incoming flow behind the third stator. This makes the two cases coherent as ϕ and ψ are always based on the incoming flow. However, when running a transient simulation this does become an ambiguous measurement. During a flow reversal the ϕ before the IGV and after the last stator would not become zero at the same time and if we were to switch between them this would cause a discontinuity in the trajectory of the operating point on the map. It is also never explicitly said which density was used to define ψ , however it seems reasonable that this would be measured at the same position of u_x .

Gamache has reported very useful data on rotating stall at low mass-flow ($\phi > 0$), zero mass-flow and low reverse mass-flow ($\phi < 0$). It is not clear, though, how the

compressor was operated in each of these regions. In the case of rotating stall with $\phi > 0$ it seems unlikely that the facility was run ‘backward’ with the flow exiting at the inlet. Most of the components are not made for reverse flow (the orifice plate, the fan or the compressor exhaust plenum) and this would have caused errors in the measurements.

Summarising, what we know is that the forward and rotating stall characteristics measured by Eastland are non-dimensionalised using the value at the inlet, the reverse characteristic reported by Gamache is non-dimensionalised using the values at the *reversed* inlet, the rotating stall reported by Gamache is non-dimensionalised using values at an unknown location.

The report from Day [58] does not clarify either the axial flow and density used. The axial velocity is calculated based on the mass-flow at the compressor outlet, obtained by measuring the throttle mass-flow and plenum pressure; this method was first described by Greitzer [33]. However how the ϕ is obtained from the discharge mass-flow and which density is used for ψ is not described in either report. During the transient simulation measured by Day ϕ becomes negative, but no discontinuity appears in the charts. Therefore, it seems reasonable to assume that the u_x considered remains the same in time, at the domain (and geometrical) inlet. The reversed characteristic reported was measured during surge cycle, and not with a dedicated experiment, therefore we can assume that it has also been measured considering the u_x at the inlet (flow outlet).

To avoid incoherent and ambiguous values during transients, it was decided in the post-processing of the model to use the inlet axial velocity and density during every regime: forward, stalled and reverse flow. This introduces some error while comparing the reverse flow characteristic with those reported by Gamache. However, at steady state there is a relation between the ψ and ϕ calculated at inlet and outlet. We define the coefficient values calculated at compressor inlet as

$$\phi_{INLET} = u_{x\ INLET} / \bar{U} \quad \psi_{INLET} = \frac{\Delta P_0}{0.5 \rho_{INLET} \bar{U}^2} \quad (5.8)$$

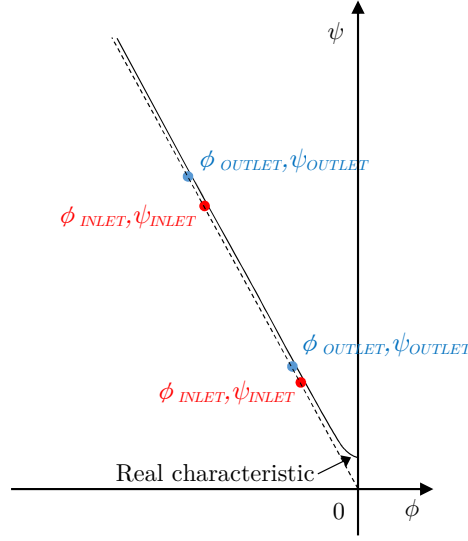


Fig. 5.3: Example of operating point calculated at domain inlet or outlet

and corresponding outlet values

$$\phi_{OUTLET} = u_{xOUTLET}/\bar{U} \quad \psi_{OUTLET} = \frac{\Delta P_0}{0.5\rho_{OUTLET}\bar{U}^2} . \quad (5.9)$$

If we define a ratio between the density at inlet and outlet

$$\Pi = \rho_{INLET}/\rho_{OUTLET} , \quad (5.10)$$

by using the continuity equation and constant area (as it is in the two cases studied) we can prove that:

$$\psi_{OUTLET} = \psi_{INLET} \cdot \Pi \quad \phi_{OUTLET} = \phi_{INLET} * \Pi \quad (5.11)$$

$$\frac{\psi_{OUTLET}}{\phi_{OUTLET}} = \frac{\psi_{INLET}}{\phi_{INLET}} \quad (5.12)$$

This means that the point on the map $[\phi, \psi]$ calculated with inlet values and the one calculated with outlet values are on the same straight line departing from the point $[0,0]$, as shown in figure 5.3. A similar demonstration can be done for the torque characteristic. The reverse flow characteristics (both pressure and torque)

observed by Gamache are parabolic but closely resemble a straight line with origin in $[0,0]$ in the range reported. The maximum value of Π observed was 1.18 at 2400 RPM at high values of ψ ; the difference reduces when considering values close to zero mass-flow.

For this reason this author decided to maintain a coherent post-processing method in which ϕ and ψ are always based on *domain* inlet, at element of index $i = 1$. The reverse characteristic reported by Gamache can still be compared to the one obtained from the model because what will be compared is the gradient and not the values themselves.

The values of u_x , ρ and P_0 used to calculate ϕ and ψ are always area-averaged. In all the cases studied, even during rotating stall, the results obtained using area-average or mass-average did not differ significantly.

Chapter 6

Results and discussion

In this chapter the results from the simulations of the new tool are reported and compared where possible against numerical (CFD) and experimental data for verification and validation purposes respectively. The first section is the verification of the 3D Euler solver obtained through comparison against a commercial CFD tool. In section 6.3 the NASA 2-stage rig is modelled in unstalled forward flow to validate the body force and the forward flow correlations. Section 6.4 reports the results from the 3-stage MIT and the modern high-speed HP compressors run in reverse flow. Section 6.5 reports the results from the rotating stall simulations for the 3-stage MIT and 4-stage C106 rigs. The compressor performance and flow features are validated and the 3D results are analysed to investigate the details of the stall cell structure. In section 6.6 the same two rigs are simulated in surge; the results are, where possible, validated with experimental measurements. The effects of plenum volume and inlet distortion on the surge cycle are investigated. Section 6.6.3 assesses the possibility of using 2D simulations instead of 3D to produce faster surge simulations. Finally section 6.1 is a grid convergence study performed on a rotating stall simulation to assess the trade-off between computational cost and accuracy.

6.1 Grid dependency study

A grid dependency study has been performed in every test case to assess the error caused by domain discretization and how close to grid convergence are the simulations run. The case here presented in detail is a full 3D, mature rotating stall on the 3-stage MIT rig, these results are presented in section 6.5.

Five different grids have been tested, reported in table 6.1. The element size h

of each grid is defined as

$$h = \left[\frac{1}{N} \sum_{i=1}^{N_X} \sum_{j=1}^{N_r} \sum_{k=1}^{N_\theta} V_{i,j,k} \right]^{1/3} \quad (6.1)$$

where $V_{i,j,k}$ is the volume of each grid element.

	Number of elements	Grid size h	Time step dt (at $t = 0$)	Computational time for 1 ms of simulated time
Case 5	$200 \times 6 \times 32 = 38400$	$1.11 \cdot 10^{-2} m$	$4.86 \mu s$	$23 s$
Case 4	$300 \times 8 \times 48 = 115200$	$0.77 \cdot 10^{-2} m$	$3.37 \mu s$	$35 s$
Case 3	$400 \times 10 \times 64 = 256000$	$0.59 \cdot 10^{-2} m$	$2.58 \mu s$	$96 s$
Case 2	$520 \times 13 \times 84 = 567840$	$0.45 \cdot 10^{-2} m$	$1.99 \mu s$	$200 s$
Case 1	$600 \times 15 \times 96 = 864000$	$0.39 \cdot 10^{-2} m$	$1.72 \mu s$	$376 s$

Tab. 6.1: Number of elements, element size, time step and computational cost of the cases used in the grid dependency study.

The case labelled 3 is the one used for the simulations in section 6.5. The other simulations are run by loading the flow-field from case 3 at steady-state, restarting the simulation on the new grid and waiting until it reaches steady-state again. To ensure the comparison is as correct as possible all the simulations are run for the same amount of time, 50 ms , and the flow-field used is the one at the end of this period. All boundary conditions (inlet, tank, throttle) are kept constant. The time step dt reported is the step at simulation's start and its size is controlled by the CFL condition, and therefore by the flow-field. The size of dt remains about constant during the simulation. The size of dt is much smaller than the characteristic convection time for this problem. Considering the smallest chord of this case (the IGV) and a maximum axial velocity of 100 m/s the minimum convection time is $\sim 0.2 ms$, about 40 times more than the dt for the coarsest grid. This ensures that the accuracy in time is sufficient to model the physical features of the problem.

Loading a previous converged solution allows to obtain a new rotating stall on finer grids with a low computational cost. Nevertheless completing the 50 ms simulation of case 1 requires over 5 hours. The stall triggering, which originally required

around 11 hours, would require ~ 2 days at this high resolution. 3D surge simulations, which already require around 48 hours for 2-3 surge cycles, would become infeasible on the finest grid.

The grid dependency study is based on the Richardson extrapolation, this is a well known method to estimate the error caused by the domain discretization and is accepted by journals specialised in numerical methods. The details of the method are well described by Celik in his 1997 paper [67] and in the guidelines for authors submitting to the Journal of Fluid Engineering [68].

The Richardson extrapolation method is used to extrapolate from the cases available the values of some variable X on an infinitely fine grid ($h \sim 0$). The extrapolation requires 3 different grids and by using different combinations of the cases in table 6.1 we can obtain more than one extrapolated value. If the cases are close enough to the asymptotic behaviour, these should coincide. Once the extrapolated value is found we can estimate the extrapolated relative error of the grid we used as

$$e_{ex} = |X_{ex} - X|/X_{ex} \quad (6.2)$$

where X_{ex} is the extrapolated value at $h \sim 0$ and X is the value obtained from the grid tested. The approximate relative error instead is the error between two grids, one finer and one coarser, defined as

$$e_{a\ fine} = |X_{coarse} - X_{fine}|/X_{fine} \quad (6.3)$$

The approximate relative error so calculated refers to the fine grid.

A grid convergence index (GCI) can also be estimated, this is a percentage error which gives a boundary within which the grid independent value of X is likely to be. The grid convergence index is defined as

$$GCI_{fine} = \frac{3 \cdot e_{a\ fine}}{r^p - 1} \quad (6.4)$$

where r is the grid refinement factor $r = h_{coarse}/h_{fine}$, and p is the apparent order of the method. The apparent order p is the order of accuracy of the method calculated

based on the cases analysed.

The Godunov method is a first order accurate in time *and* space. When the grid element size h is reduced the accuracy will increase because of the finer space discretization. However since the dt is calculated based on the CFL condition, a reduction in element size also makes the time discretization finer, see table 6.1. Since the case we study is a transient, we expect the refining in time discretization to also have an impact on the results. As both the effects will occur when we refine the grid, we are not expecting the apparent order of accuracy p to be 1, but higher.

The two critical variables X analysed are ϕ and ψ , their values are listed in table 6.2 and plotted in figure 6.1.

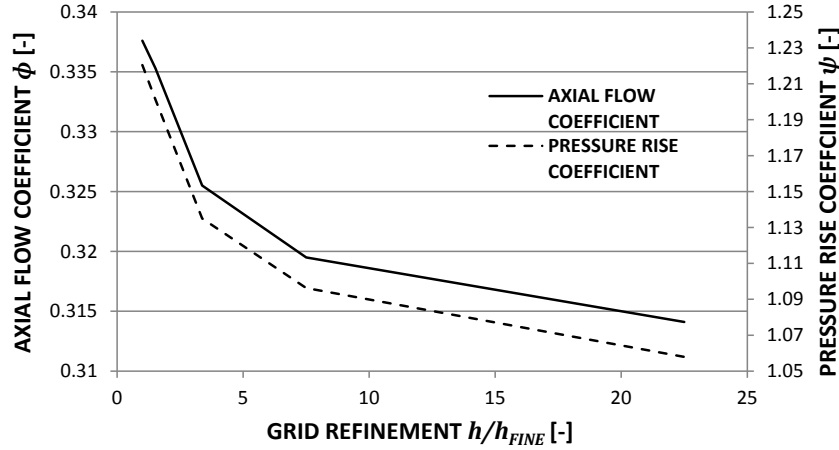


Fig. 6.1: Values of axial flow coefficient and pressure rise coefficient with the increasing grid refinement.

The combinations studied are cases 2,3,4 and cases 1,3,5; the results are listed in table 6.2. The values of ϕ and ψ extrapolated are very similar for both combinations. These extrapolated values are those at $h \sim 0$, and seem in accordance with the trends shown in figure 6.1. If the combination 1,2,3 is used both the extrapolated values become much lower ($\phi = 0.34, \psi = 1.26$). This is reasonable as those 3 points seem to indicate that the trend becomes flatter for grids finer than case 1. However the grid sizes of cases 1 and 2 are too close to each other and the ratio r is lower than the minimum value of 1.3 suggested by Celik [68], therefore this combination is not considered reliable and discarded. The e_{ex} , e_a and GCI_3 reported are calculated for case 3. The approximate relative error e_a requires a finer and a coarse grid to be

	ψ		Φ	
Cases	2,3,4	1,3,5	2,3,4	1,3,5
X_{fine}	1.2012	1.2204	0.3353	0.3376
$X_{case\ 3}$	1.135	1.135	0.3255	0.3255
X_{coarse}	1.0963	1.0579	0.3195	0.3141
p	2.03	2.01	1.86	1.84
X_{ex}	1.294	1.288	0.350	0.349
e_{ex}	12.3 %	11.9%	7.18 %	6.65 %
e_a	3.41%	6.79%	1.84%	3.50%
$GCI_{case\ 3}$	5.94%	3.33%	3.6%	2.00 %
GCI_{fine}	9.6%	6.97%	5.93%	4.04%

Tab. 6.2: Results of the grid dependency study.

calculated, in this study the fine grid considered is case 3, and the coarse is either 4 or 5.

In figure 6.1 we can observe that both the variables have a particular trend, with a sharp increase between cases 2 and 3. As a result the approximate errors, e_a , between the coarse grids (4-5) and grid 3 are very low compared to the extrapolated errors e_{ex} ; the same is true for the GCIs. For comparison the GCI values of the fine grids (2 or 1) are also reported and these are consistently higher than those of case 3. An increase in error for a finer grid is unreasonable, for this reason the GCI_3 value are discarded and it should be considered that the GCI of cases 3 should be greater than GCI_{fine} . The best estimate of the grid discretization error of case 3 is then the extrapolated relative error e_{ex} .

The apparent order calculated during this analysis is about 1.9; as expected, it is higher than the theoretic first order of accuracy in space and time of the scheme.

In conclusion, from this analysis it was underlined how the results obtained in the simulations are relatively far away from the result which would be obtained on an infinitely fine grid. In the case studied the results should have up to a 12% error band due to grid discretization error. However as described before the grid size of

case 3 has been chosen as a trade off between accuracy of the results and feasibility of the simulation.

It is important to remember that in this code viscosity is not modelled and the only (numerical) viscosity present is caused by the diffusion implicit in the scheme. As the grid becomes finer the numerical diffusion also reduces and the flow-field modelled becomes more and more close to inviscid. Therefore, a grid independent solution cannot be reached as the viscosity of the flow modelled is a function of the grid. Furthermore the flow-field modelled on an infinite grid, which is completely inviscid, is not necessarily closer to the real physical flow than the one with some viscosity introduced by a coarse grid.

6.2 Three-dimensional Euler solver validation

The first step of the new tool development is the validation of its three-dimensional Euler solver. To validate only the solver an empty duct is modelled and the results are compared with a simulation performed using ANSYS FLUENTTM. The geometry used is the annulus from the experimental rotor R37 with the blading removed [65]. In order to obtain a three-dimensional flow a constant pressure distortion is introduced at the inlet on a 90° sector. The pressure distortion is chosen small to avoid flow reversal and it causes only a redistribution from the undistorted flow towards the distorted sector, as shown in figure 6.2. This pressure distortion is not representative of any real situation, but is only a step change in inlet conditions chosen for simplicity of model set-up. The shape of the duct and of the distorted sector are shown in figure 6.3. Three lines are also presented which are used to compare the profiles of pressure, axial velocity and circumferential velocity. Line A is at 90° from the distorted sector, line B is at the center of the sector and line C is on the edge. The parameters of the simulation are reported in table 6.3.

The ANSYS FLUENT model solves the compressible Euler equations (inviscid), a density based solver is used: the Implicit Roe-FDS (Flux difference splitting). ‘Pressure inlet’ boundary conditions are used at inlet in both sectors, ‘pressure outlet’ conditions are used at the outlet, the flow direction at inlet is imposed normal to

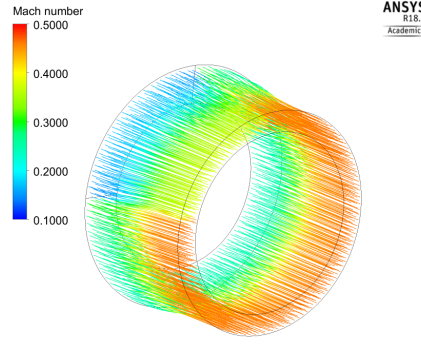


Fig. 6.2: Three-dimensional flow modelled using ANSYS FLUENT.

the surface. These boundary conditions are chosen as they have the same properties as those developed for the new code.

The mesh used in FLUENT is unstructured, automatically generated with tetrahedral elements. A large number of elements is chosen, about a million, for both the simulations. The number of axial, radial and circumferential elements for the new model has been chosen to maintain an aspect ratio of the volumes to be ~ 1 :

$$dx \sim \overline{dr} \sim d\theta \cdot r \quad (6.5)$$

in which $\overline{dr}$ is an average of the radial size of the volumes.

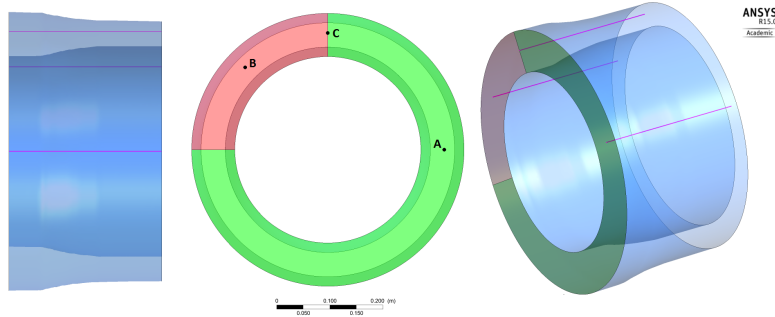


Fig. 6.3: Annulus used in the Euler solver validation. Distorted sector and lines used for comparison are highlighted.

A grid convergence study is performed on this test case, in a similar fashion to what described in section 6.1. Two grids are tested: one coarser ($63 \times 18 \times 388$) and one finer ($106 \times 31 \times 660$); the variables X considered are the mass-flow $\dot{m}$, the average inlet static pressure $\overline{P}_{IN}$, the average outlet total pressure $\overline{P}_{OUT}$. To ensure

FLUENT SIMULATION		THROUGH-FLOW CODE	
Mesh type	Unstructured	Axial elements	82
Element type	Tetrahedral	Radial elements	24
		Circumferential elements	508
N. elements	1000000	N. elements	999700
Boundary conditions			
Inlet clean Total pressure [Pa]			101000
Inlet clean Total temperature [K]			288
Inlet distorted Total pressure [Pa]			96000
Inlet distorted Total temperature [Pa]			288
Outlet static pressure [Pa]			87000

Tab. 6.3: Simulation parameters for the Euler solver validation.

the three-dimensional features of the flow are correctly modelled the circumferential velocity is also considered; the value at outlet on Line C is used as representation. The values of these variables on an infinite grid are extrapolated using Richardson's method and presented in table 6.4 with corresponding relative errors e_{ex} and GCI index; the relative errors at different levels of grid refinement are also displayed in figure 6.4. The extrapolated error for the grid used have a maximum of 1.17% on the mass-flow, with a correspondent maximum GCI of 1.49%.

The results from the two models are compared along the three lines, in figures 6.5 and 6.6. The axial position x' is non-dimensionalized using the length of the duct, the pressure P' is non-dimensionalized using a reference pressure of 101325 Pa.

$$P' = P/101325 \quad (6.6)$$

The results along the A and B lines are matched very closely and some mismatch exists only the edge of the distorted sector.

The difference between the results from the new tool and the high-fidelity CFD

Variable X	Mass-flow $\dot{m}$ [kg/s]	Inlet pressure $\bar{P}_{IN}$ [kPa]	Outlet pressure $\bar{P}_{0OUT}$ [kPa]	Circ. velocity $u_{\theta OUT C}$ [m/s]
X_{coarse} $63 \times 18 \times 388$	9.934	96.14	99.38	8.137
$X_{case\ tested}$ $82 \times 24 \times 508$	9.9795	96.10	99.51	8.156
X_{fine} $106 \times 31 \times 660$	10.0108	96.08	99.6	8.167
X_{ex}	10.0981	96.02	99.87	8.184
$e_{ex\ case\ tested}$	1.17 %	0.08%	0.36 %	0.33%
$GCI_{case\ tested}$	1.49 %	0.11%	0.46%	0.42%

Tab. 6.4: Grid convergence study for the 3D Euler solver case.

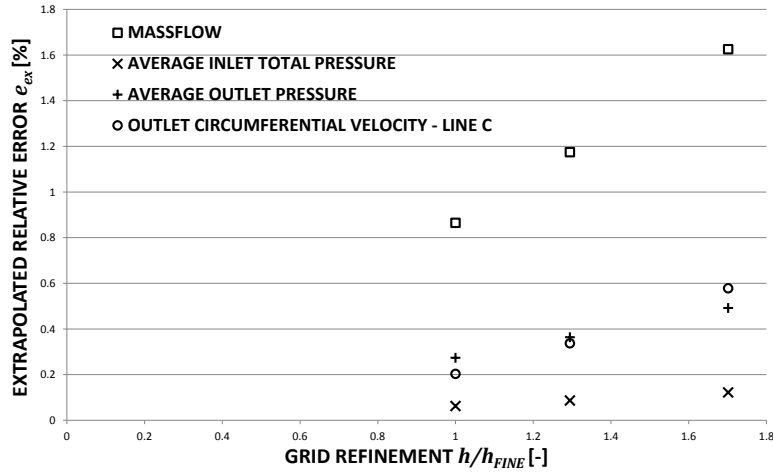


Fig. 6.4: Extrapolated errors from the grid convergence study on the Euler solver test case.

is presented as a relative discrepancy, defined as:

$$DIS = \left| \frac{X_{TOOL} - X_{CFD}}{X_{CFD}} \right| \quad (6.7)$$

where X is the generic results considered.

The maximum relative discrepancy is the circumferential velocity profiles on line

C, going up to $\sim 12\%$ at outlet. On line A and B the discrepancy remains lower than 1% on all the variables. The presence of a discontinuity at the sector edges causes a jump in the pressure values at the inlet. This can be seen in figure 6.6, the profiles of inlet pressure at radius $r = 0.22$ are matching closely but a difference exists at the sector's edges where the new model has larger peaks of lower pressure (these exists also in FLUENT but are minimal). This is probably occurs because the discontinuity is modelled slightly differently by the two solvers. Results of integral variables ($\dot{m}$, P , P_0) are compared in table 6.5 and are all matched with discrepancy lower than 1%.

	MODEL	CFD	Relative discrepancy
Mass-flow [kg/s]	9.98	10.06	0.8%
Outlet P'_0 [Pa]	0.9821	0.9820	0.07 %
Inlet P' [Pa]	0.9484	0.9486	0.01%

Tab. 6.5: Comparison of integral results from the new tool and ANSYS FLUENTTM.

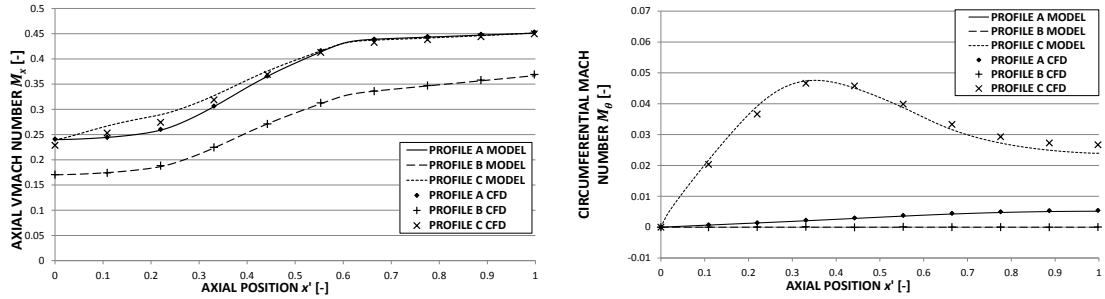


Fig. 6.5: Comparison of axial and circumferential velocity modelled with the new tool and ANSYS FLUENTTM.

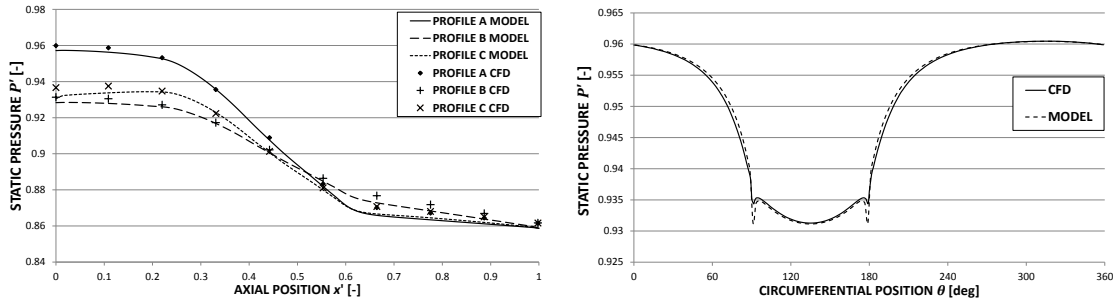


Fig. 6.6: Comparison of static pressure modelled along the three lines (left) and at constant radius at inlet (right).

6.3 Forward flow validation

The modelling of a steady state in forward flow is validated reproducing the compressor map of the 2-stage NASA rig [61]. Reproducing a compressor map allows to validate the correct functioning of the body forces implemented, the empirical correlations for forward flow and the plenum/throttle model.

The 50% (8031 RPM) and 70% (11243 RPM) speed-lines are reproduced and compared to the experimental data. The 80% and higher speed-lines were not reproduced as at those speeds the compressor operates in the transonic region and the empirical correlations are not suitable to model this regime.

The simulations are run in 2D on a $800 \times 20 \times 1$ domain. A grid convergence study was performed reported in table 6.6. The variation of extrapolated relative error with the grid refinement is presented in figure 6.7.

With respect to other grid size used in similar through-flow codes (for example the grids used by Longley [41]) the grid used here has a much finer resolution along the axial direction. This was chosen as the larger number of elements along the pseudo-planes makes the body-force method more stable during transient and accurate. This is because during transient (and especially reverse flow) the body forces will have to impart a large instantaneous turning to the flow and a larger number of elements allow to distribute better the forces and with a better resolution. The impact of the aspect ratio on the result was tested by running the simulation with different number of radial elements: 800×30 and 800×10 . By comparing the 3 grids, the obtained mass-flow varies of less than 0.4%, pressure and temperature ra-

Variable X	Mass-flow $\dot{m}$ [kg/s]	Pressure ratio PR [-]	Temperature ratio TR [-]
X_{coarse} $400 \times 10 \times 1$	16.3549	1.1689	1.064
$X_{case\ tested}$ $800 \times 20 \times 1$	16.6153	1.1726	1.0629
X_{fine} $1200 \times 30 \times 1$	16.7247	1.1744	1.0621
X_{ex}	16.804	1.1761	1.06
$e_{ex\ case\ tested}$	1.12% %	0.29%	0.27 %
$GCI_{case\ tested}$	1.42% %	0.37%	0.35%

Tab. 6.6: Grid convergence study for the the 2-stage NASA rig case

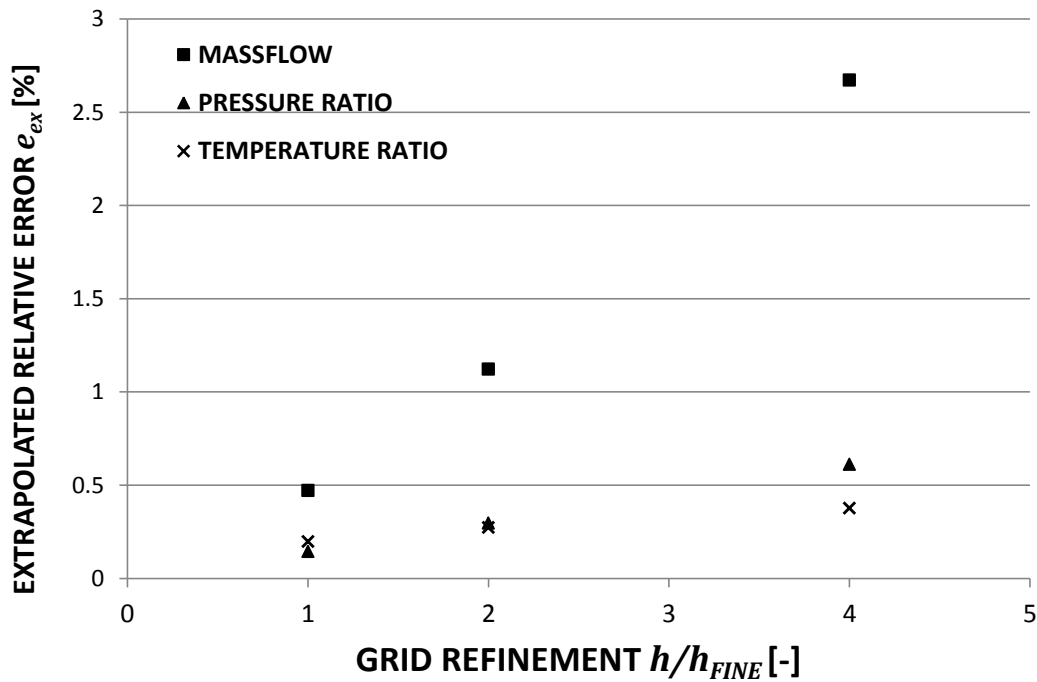


Fig. 6.7: Extrapolated errors from the grid convergence study for the 2-stage NASA rig case.

tio vary of less than 0.1% Therefore the results do not depend on the element aspect ratio in the meridional plane and in the radial direction the grid is well converged.

The method described in section 4.9.3 is used for the acceleration to full speed and creation of the first point of the characteristic. Subsequent throttle reductions are used to move along the characteristic. The correlations used are those described by Aungier, which have been tuned on the data from this rig.

In figures 6.8 and 6.9 the modelled pressure and temperature maps are compared with those from experimental data. The mass-flow used in the maps is corrected with the average value of the inlet pressure and temperature

$$\dot{m}' = \dot{m} \cdot \frac{\sqrt{T_{0\,IN}/T_{REF}}}{P_{0\,IN}/P_{REF}} \quad (6.8)$$

where the reference conditions are 288 K and 101325 Pa . The inlet conditions profiles used in the model are the same as in the experiment so that the results can be compared.

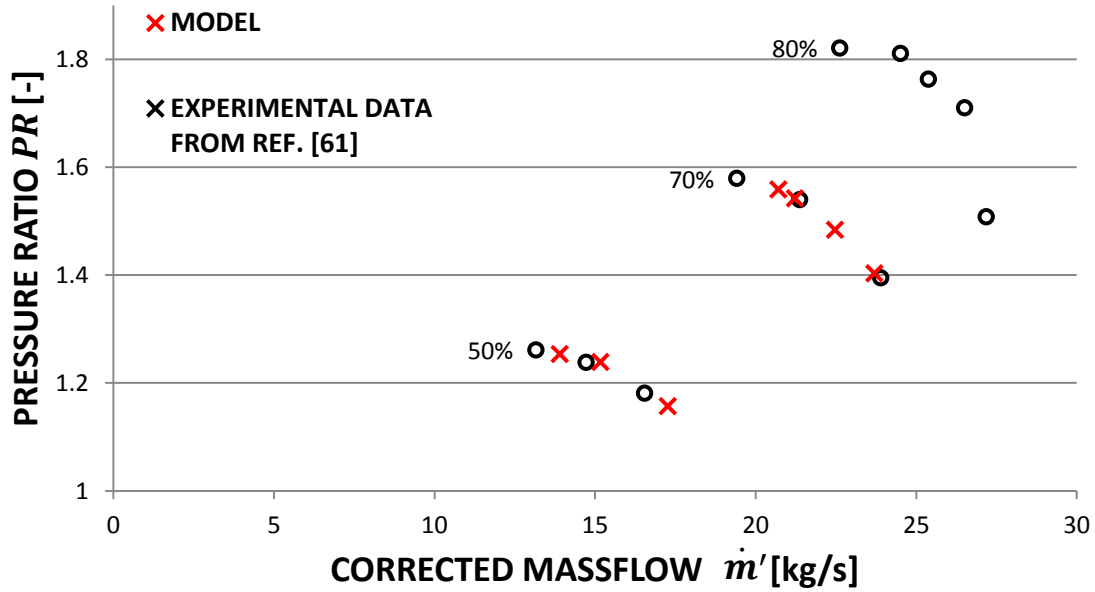


Fig. 6.8: Comparison of experimental and modelled compressor map of the NASA 2-stage rig, experimental data from Urasek et al. [61].

Both the characteristics are reproduced satisfactorily with maximum errors of 2.5% in the pressure ratio and 10.8% in the temperature rise (1.8 K error on a 17 K rise, at the lowest point). The maximum pressure rise estimation is too conservative and both the lines have a lower surge margin than the experimental data. This suggests a better criteria to detect stall should be researched and implemented in

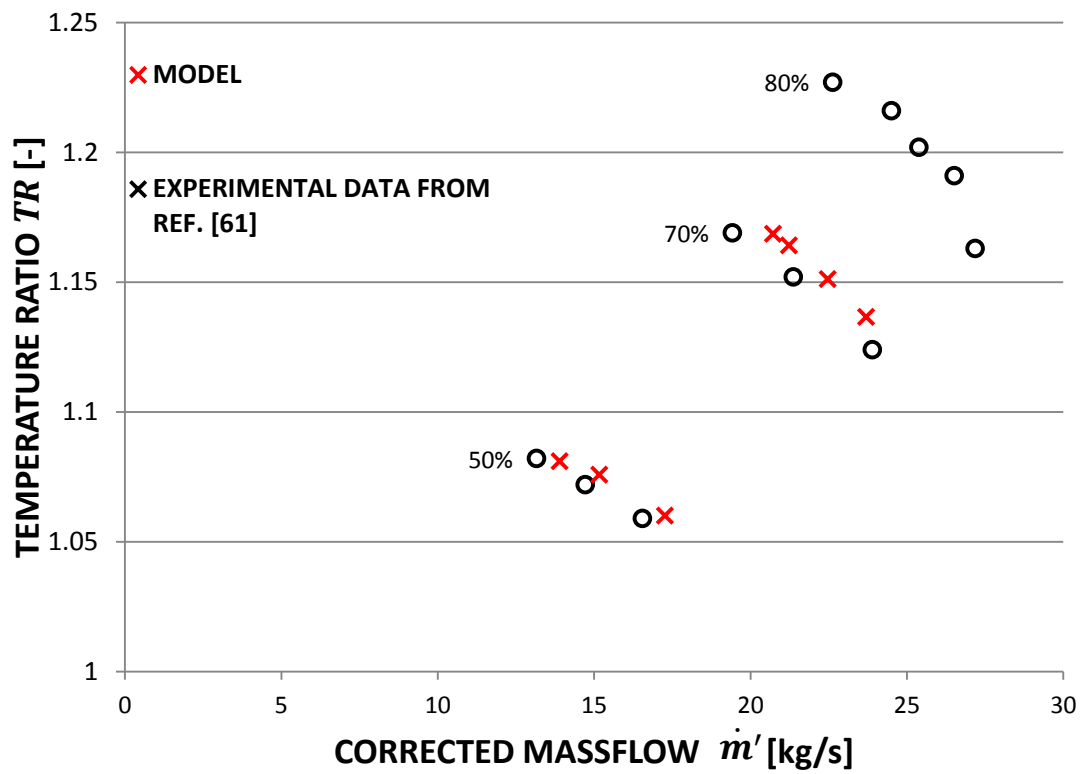


Fig. 6.9: Comparison of experimental and modelled temperature ratio map of the NASA 2-stage rig, experimental data from Urasek et al. [61].

the code in the future.

6.4 Reverse flow validation

6.4.1 3-stage MIT rig

Reverse flow validation was performed on the 3-stage MIT rig, comparing with the experimental reverse flow characteristic reported by Gamache [57].

The model used is a two-dimensional version of the one used for the rotating stall simulation reported in section 6.5, the grid is therefore $400 \times 10 \times 1$. The grid convergence was tested in 3D and reported in section 6.1.

Data is reported for 3 different rotational speeds: 1170, 1800, 2400 RPM. On the $\psi-\phi$ map the 3 characteristics are overlapping. In this thesis the experimental points are not divided by speeds but drawn as part of a single line. The characteristics at the 3 different speeds are reproduced and compared as shown in figure 6.10.

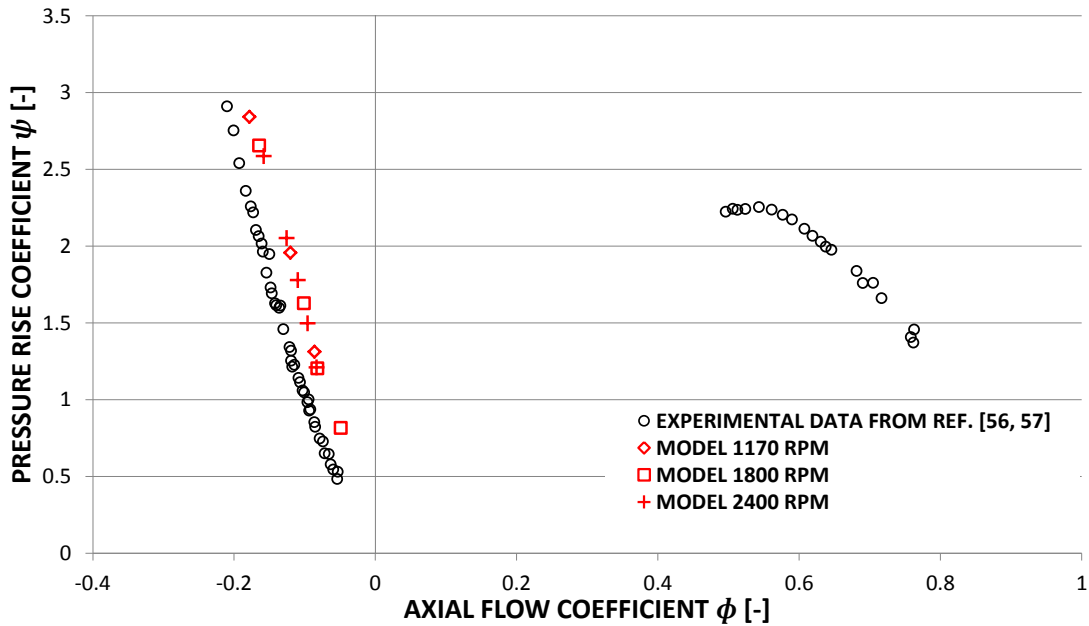


Fig. 6.10: Reverse characteristic validation on the 3-stage MIT rig, experimental data from Eastland [56] and Gamache [57].

The 3 velocities display the same overlapping as seen experimentally, although the simulated characteristics have a lower reverse mass-flow. The trend is well reproduced, as well as the gradient of the characteristic with an error of $\sim 9\%$. The simulated characteristics do not reach the lowest values of experimental ψ , as

the compressor recovers too early because of the higher pressure rise. The recovery however occurs at about the same ϕ as reported.

The temperature characteristics for reverse flow were not provided by Gamache, but the characteristics of torque coefficient are available instead. Like the pressure characteristics also the torque characteristics are coinciding for the 3 different rotational speed studied. The simulated compressor is assumed adiabatic and the only change in total temperature is due to the term W_{EX} in the rotors. This value is directly dependent on the torque

$$W_{EX} = F_T \cdot U = \mathcal{T} \cdot \Omega \quad (6.9)$$

where Ω is a constant value for the compressor. Therefore a match of the steady torque characteristics would mean that the temperature rise through the compressor is being simulated correctly.

The comparison of the torque characteristic in reverse flow regime is shown in figure 6.11.

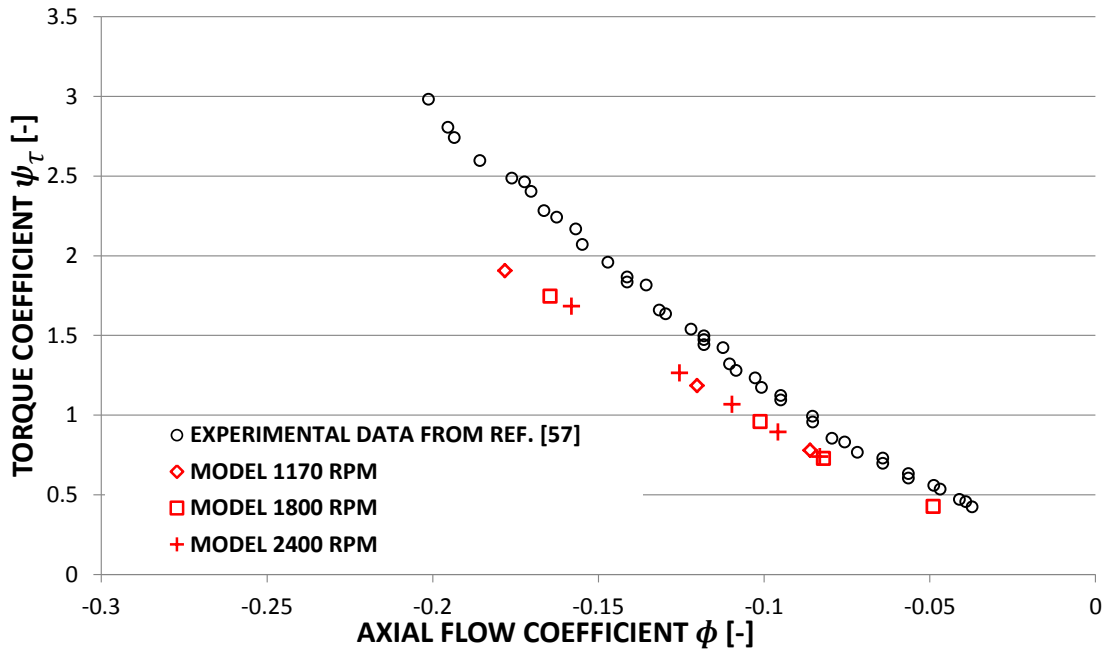


Fig. 6.11: Reverse flow torque characteristic validation for the 3-stage MIT rig, experimental data from Eastland [56] and Gamache [57].

The torque appears to be slightly under-predicted by the current model by an

average 20%. A similar under-prediction was also reported by Longley (15% less) in his simulation [41] and also by the 2D mean line prediction performed by Gamache itself (30% less). The reported data are linear from zero flow until a $\phi = -0.08$, at higher negative flow coefficient the steepness of the curve increases. The curve of the torque coefficient from the simulations instead has a constant direction and therefore diverges at high reverse flow.

The span redistribution of the flow during reverse flow is also reported qualitatively by Gamache, figure 6.12. The profiles shown in the picture are those of the ϕ , normalised over the mean value at that axial position.

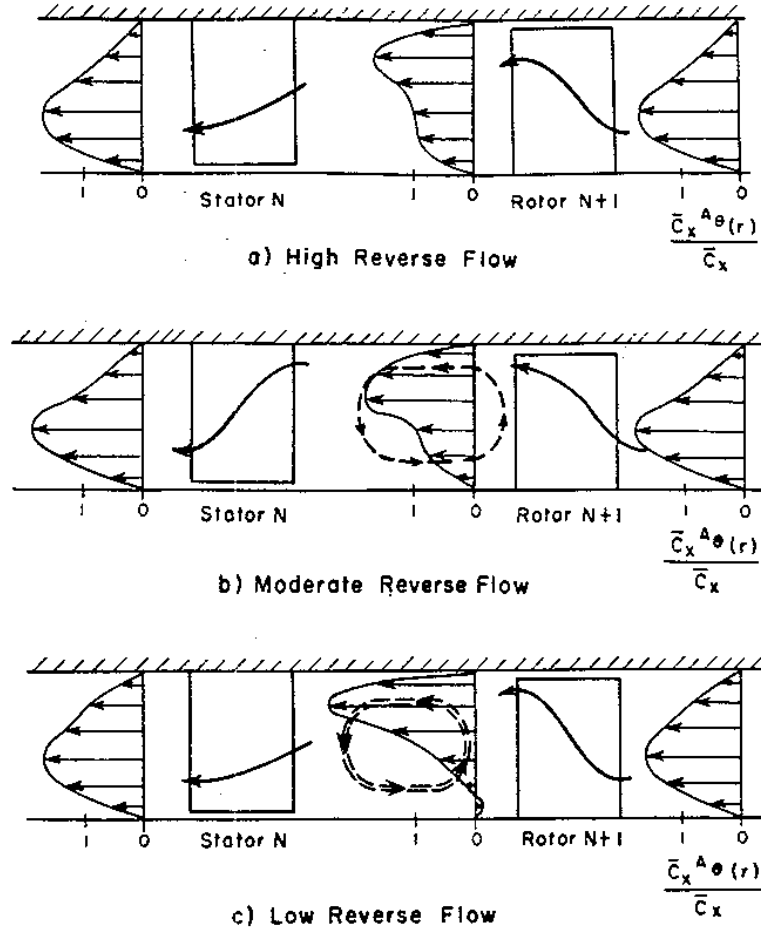


Fig. 6.12: Span distribution of fluid in the meridional plane during reverse flow [57].
The flow coefficients reported are a) $\phi = -0.19$, b) $\phi = -0.10$, c) $\phi = -0.05$.

The results from the modelled reverse flow are post-processed in a similar fashion to compare the radial distribution. In figures 6.13 and 6.14 the flow through the 1st stage stator and the 2nd stage rotor is visualised as arrows on the meridional plane.

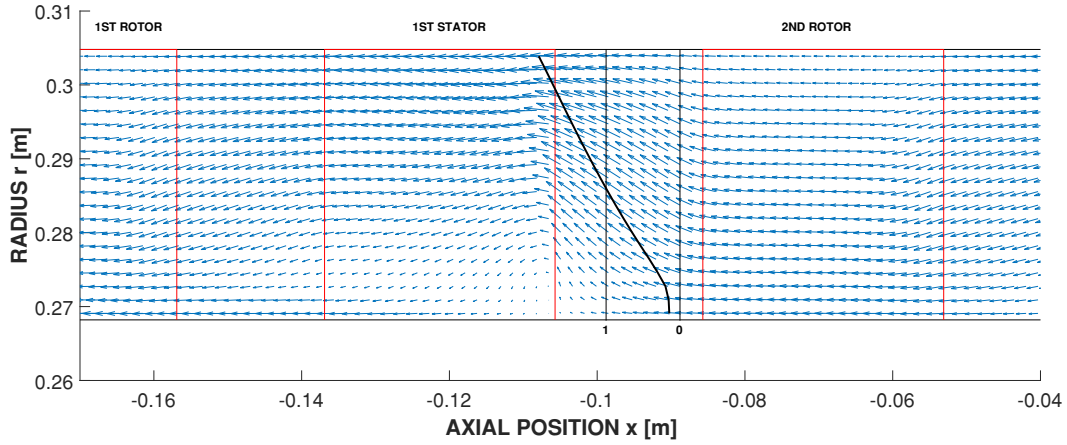


Fig. 6.13: Modelled flow-field in the meridional plane; with blades shape over-imposed in red and axial flow coefficient profile over-imposed in black. $\phi = -0.1$.

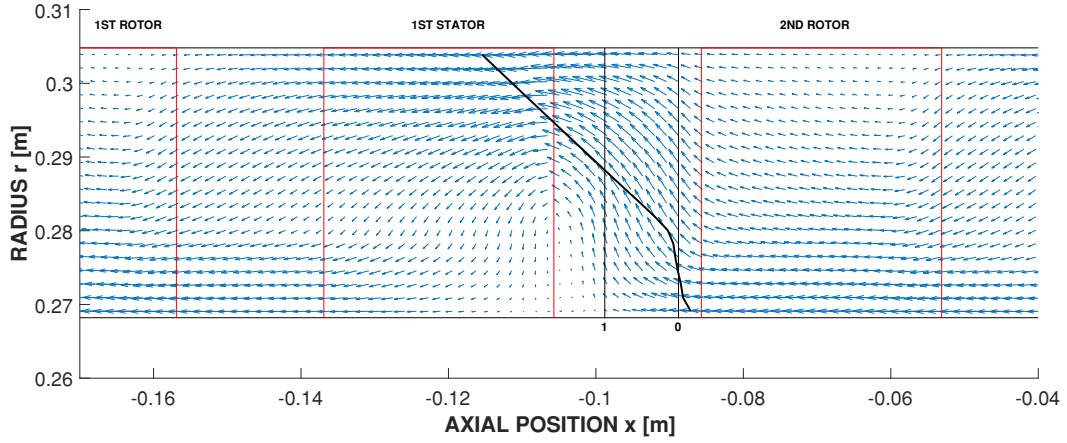


Fig. 6.14: Modelled flow-field in the meridional plane; with blades shape over-imposed in red and axial flow coefficient profile over-imposed in black. $\phi = -0.05$.

Over-imposed in black are the profiles of ϕ , normalised over their mean value at that axial position. Figure 6.13 is at $\phi \approx -0.1$ corresponding to case b); figure 6.14 is at $\phi \approx -0.05$, corresponding to case c). In the post-processing of the simulated solution the axial position at which ϕ is probed is $x = -0.103$, immediately behind the stator trailing edge.

The structure of the flow distribution reproduces what was observed experimentally, with a high radial flow towards the tip downstream of the rotor and the opposite redistribution downstream of the stators. The profile coefficients have similar trends to the experimental data. At $\phi = -0.1$ there is a higher reverse flow closer to the tip than the hub, and the profile becomes more extreme at $\phi = -0.05$ with even higher flow at tip and positive flow at the hub. It is evident that the wall

boundary layer has an important effect on the experimental flow profile. In fact this is not modelled by the Euler solver and the profiles from the code have maximum velocities close to the walls, unlike the experimental ones. The re-circulation downstream of the rotor reported from Gamache at low reverse flow is reproduced in the corresponding simulated flow-field.

6.4.2 *Modern high-speed HP compressor*

The reverse flow treatment is also tested on a high-speed, 6-stage modern HPC compressor. The compressor was previously modelled using ANSYS CFX for 2 rotational speeds by Pawsey [10], during his project within the Cranfield UTC. A stage-stacking technique (SST) was also developed and tuned using the CFD data, allowing to create the full range of reverse flow characteristics. This data are used to verificate the capability of the current code to model a modern HPC compressor.

The body force formulation described in section 4.3 is used but it could not simulate properly the characteristics at higher speeds. The cause is the correction proposed by Brand which has not been adapted to transonic conditions and becomes singular at $M \sim 1$. To simulate also the higher speeds a simpler formulation is used, with only the centripetal and the corrective forces F_{CORR} . Using this formulation in case of high Mach number the turning force is under-estimated, however as the flow mis-aligns the corrective force increases and limits how much the direction diverges. To increase the response of the corrective force coefficient K_{CORR} was increased to 20 to minimise the misalignment at the flow outlet.

The simulations were run axi-symmetrically (2D), the boundary conditions are set constant and the static pressure at domain inlet (flow outlet) is always 100 kPa . The total temperature at the outlet (flow inlet in reverse) is set at 288 K while the total pressure changes between simulations and is used to create the points on the characteristics.

The mesh used has 1200 axial, 20 radial and 1 circumferential elements. A grid convergence study has been performed for this set-up; as the range of operating conditions vary greatly, two cases are analysed: one at low speed $0.2\text{ }NRT^*$ and one at high speed $0.8\text{ }NRT^*$. The results are presented respectively in tables 6.7 and

6.8 and the extrapolated relative errors are presented in figure 6.15. The grid used is very well converged for the low speed case with maximum GCI of 0.79% on the mass-flow; however, this resolution is necessary at higher speed where the grid is further away from convergence. Overall, with the grid selected the maximum GCI observed is $\sim 5\%$.

Variable X	Scaled corrected mass-flow $\dot{m}' \cdot K_{SCALE X} [-]$	Scaled pressure ratio $PR \cdot K_{SCALE Y} [-]$	Scaled temperature ratio $TR \cdot K_{SCALE Y} [-]$
X_{coarse} $600 \times 10 \times 1$	-1.1008	0.5059	0.3389
$X_{case\ tested}$ $1200 \times 20 \times 1$	-1.0986	0.5051	0.3401
X_{fine} $1560 \times 26 \times 1$	-1.0969	0.5049	0.3404
X_{ex}	-1.0930	0.5042	0.3414
$e_{ex\ case\ tested}$	0.51% %	0.17%	0.40%
$GCI_{case\ tested}$	0.79% %	0.35%	0.61%

Tab. 6.7: Grid convergence study for the modern high-speed HP compressor, case at low speed $0.2 NRT^*$

The results are shown in figures 6.16 to 6.19. The continuous lines represent the results from the stage stacking technique, while the black dashed lines are the CFD results at speeds $0.4NRT^*$ and $1.0NRT^*$. In figures 6.16 and 6.17 are reported the results from the model with the simpler body force method, labelled ‘MODEL’. In figures 6.18 and 6.19 are reported the results from the model with the body force corrected for Mach number, labelled ‘MODEL 2’. The latter can not reach speeds higher than $0.6NRT^*$, as the insurgence of transonic conditions makes the body forces unstable. All the simulated results have a lower limit of pressure ratio under which the compressor recovers, the lines from the stage stacking technique instead were extrapolated until the zero mass-flow.

Variable X	Scaled corrected mass-flow $\dot{m}' \cdot K_{SCALE X} [-]$	Scaled pressure ratio $PR \cdot K_{SCALE Y} [-]$	Scaled temperature ratio $TR \cdot K_{SCALE Y} [-]$
X_{coarse} $600 \times 10 \times 1$	-0.4886	0.5541	0.7577
$X_{case\ tested}$ $1200 \times 20 \times 1$	-0.5077	0.5381	0.7708
X_{fine} $2400 \times 40 \times 1$	-0.5171	0.5290	0.7734
X_{ex}	-0.5291	0.5168	0.7742
$e_{ex\ case\ tested}$	4.13% %	0.43%	4.065%
$GCI_{case\ tested}$	4.96% %	0.54%	4.60%

Tab. 6.8: Grid convergence study for the modern high-speed HP compressor, case at high speed $0.8 NRT^*$.

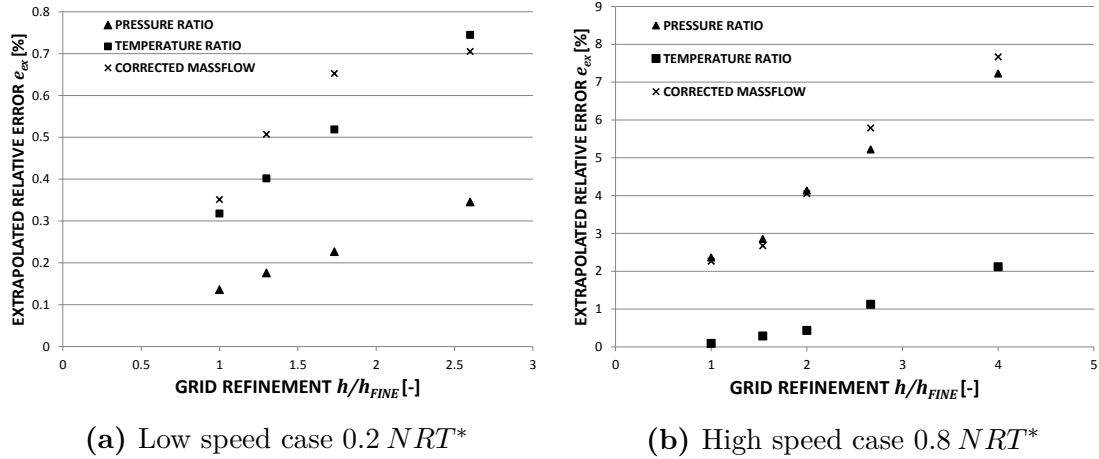


Fig. 6.15: Variation of extrapolated relative error with grid refinement in the modern high-speed HP compressor case.

The results with simple forces ('MODEL') follow closely the $0.4NRT^*$ line and CFD data; the results also follow the correct trends for higher speed-lines but underestimate the mass-flow. The results obtained from the Mach corrected forces ('MODEL 2') follow very closely the CFD data at $0.4NRT^*$, but quickly diverge at higher speeds with the $0.6NRT^*$ results being closer to the $0.8NRT^*$ line. As

explained in section 4.7 the correlation used for deviation is not applicable to compressible flows and this might justify the bad matching with both methods. At $0.4NRT^*$ instead the relative Mach number remains generally lower than 0.6 in all the blade rows. As a result the correction proposed by Brand does not influence much the body forces and the two models have very similar results. At $0.2NRT^*$ the results diverge from the SST line. However this line was not verified with CFD but extrapolated, and with the current data it is not possible to determine which low order prediction method (current tool or the SST) is the most correct one.

On the temperature map the matching is very accurate at lower speeds while at higher NRTs it becomes less accurate. In particular, the shape of the curves appears to be well reproduced but shifted at lower mass-flow (as also seen in the pressure ratio map), and with a constant under-estimation of the temperature ratio. The two formulations of body forces have very similar results in terms of temperature ratio, although with a shift in mass-flow predicted.

In the current model the gas is considered thermally perfect, but at high speeds the temperatures reached are so high ($> 800K$) that this assumption can not be held true anymore. At high temperatures the specific heat increases requiring more energy to raise the temperature of one degree. This observation suggests that the current model where C_P is constant would over-estimate the temperature ratio (for the same energy input, the temperature increases more), however the opposite trend is observed. This seems to indicate that the constant specific heat is not the only problem and some other effect concurs to the difference in simulated temperature ratio from the data.

Summarising, the tool was applied to a 6-stage high-speed modern compressor and matched satisfactorily the low speed CFD data. At higher speeds the code is not applicable and the data was matched poorly. Overall the tool was able to reproduce the correct trends in both pressure and temperature ratio over a large range of speeds and mass-flow.

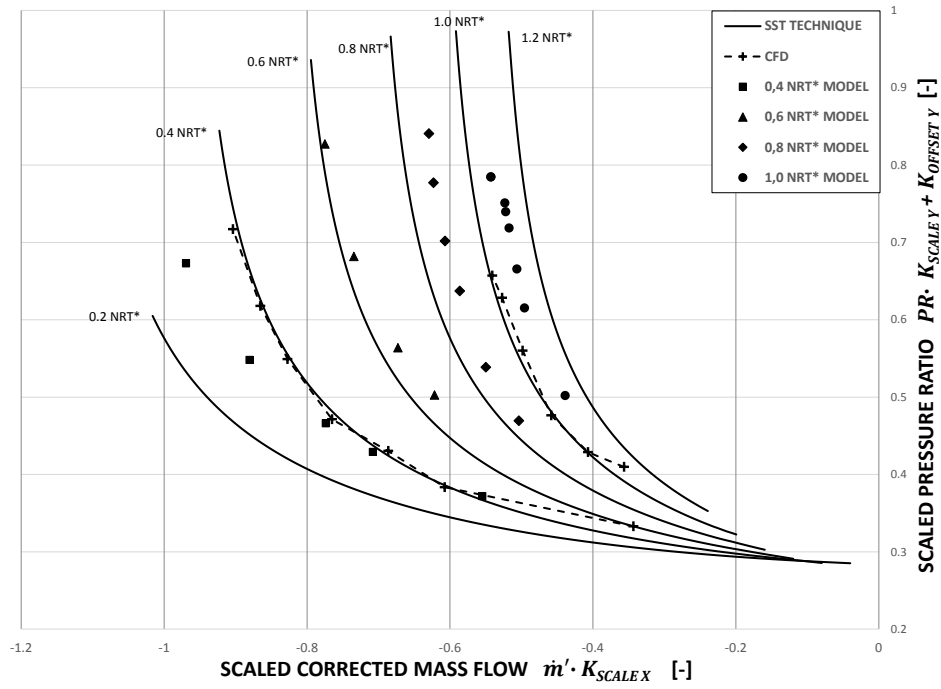


Fig. 6.16: Reverse flow characteristics validation on the 6-stage high-speed HPC compressor, using a simplified body-force method.

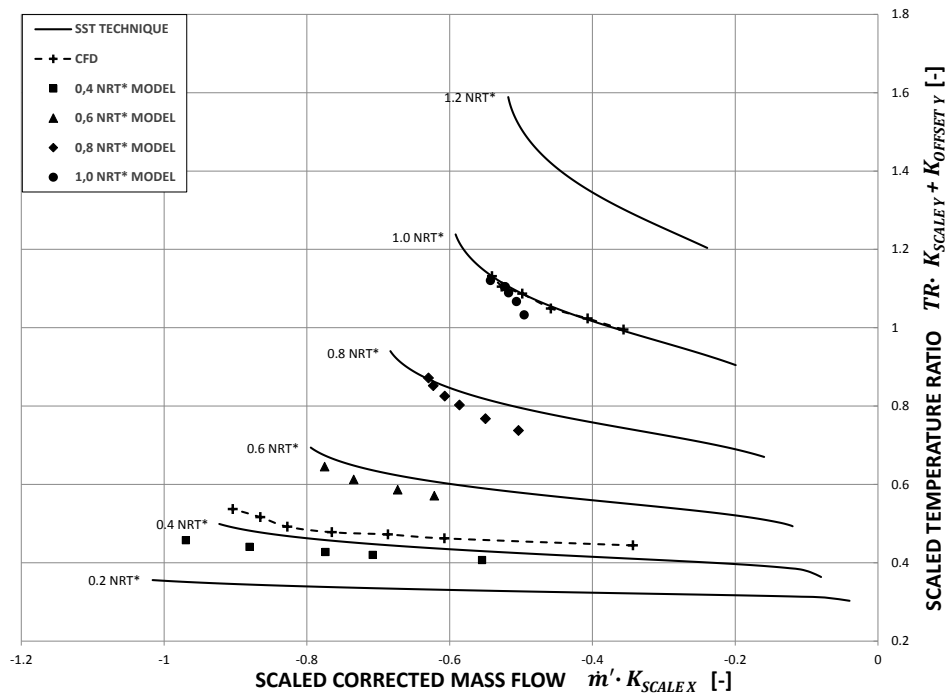


Fig. 6.17: Reverse flow temperature characteristics validation on the 6-stage high-speed HPC compressor, using a simplified body-force method.

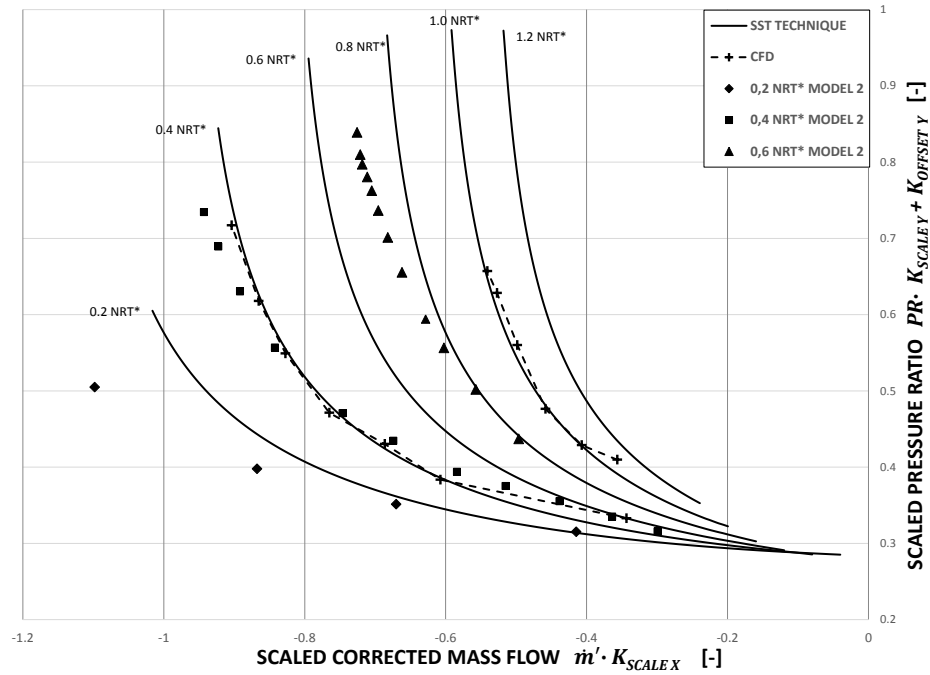


Fig. 6.18: Reverse flow characteristics validation on the 6-stage high-speed HPC compressor, using the body force method with correction for compressibility.

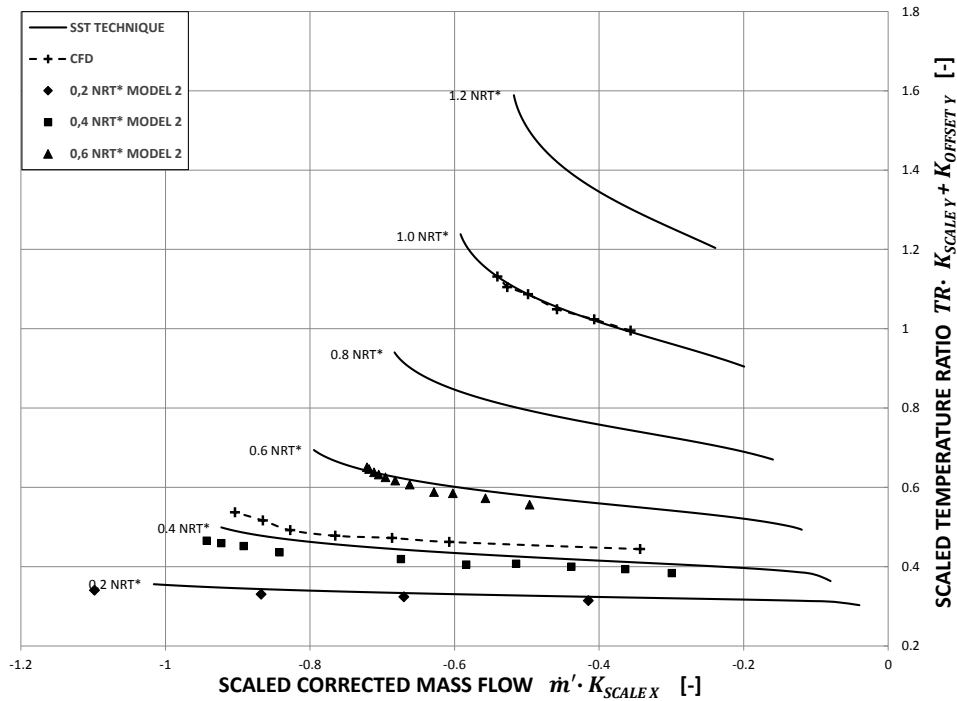


Fig. 6.19: Reverse flow temperature characteristics validation on the 6-stage high-speed HPC compressor, using the body force method with correction for compressibility.

6.5 Rotating stall validation

The rotating stall modelling is validated on the 3-stage MIT rig and the 4-stage C106 rig from Cambridge. The method described in section 4.9.3 was used in both validation cases to speed up the creation of the rotating stall characteristic.

In both cases a temporary distortion at the inlet is used to create some asymmetry and allow the compressor to develop a rotating stall. Two patterns are used during rotating stall and surge simulations, as shown in figure 6.20. Pattern A has a small perturbation which does not change the average inlet P_0 but just introduces a background asymmetry. Pattern B instead creates a strong reduction in flow locally and can also act as a trigger to stall itself. The time also changes: Pattern A lasts only 4 ms while Pattern B lasts 10 ms.

The two patterns described are chosen arbitrarily and are not representative of realistic inlet distortion. In order to reduce the computational cost of the simulation the distortions are impulsive and last only few milliseconds; as the distortion is a single impulse, a large magnitude of pressure reduction is used. The magnitude of the distortion can be quantified using various distortion descriptors [66], however these are used for distortion which exist over a relatively longer period of time and are of limited use to quantify an impulsive distortion. To compare the presented distortion patterns, the total pressure distortion descriptor DC60 is used, defined as:

$$DC60 = \frac{\overline{P_0}|_x - \overline{P_0}_{60}}{\overline{q}|_x} \quad (6.10)$$

where $\overline{P_0}|_x$ is the average total pressure on an annulus section at position x and $\overline{P_0}_{60}$ is the lowest average total pressure on a 60° sector at the same position. The descriptor is non-dimensionalized using the average dynamic head on that section, $\overline{q}|_x$.

The values of DC60 normally observed experimentally are well below 1; for example, in complex S ducts intake values of DC60 at compressor inlet do not exceed ~ 0.6 [73]. In the few milliseconds in which they are applied, the two patterns proposed have a DC60 of respectively 2.2 (Pattern A) and 5.06 (Pattern

B), much larger than the realistic values. In case of Pattern B the distortion is so large that locally the flow is instantaneously inverted and the distorted intake sector becomes an outlet. Despite the large value of DC60, because of its small duration the Pattern A distortion does not cause a strongly a-symmetric flow, see section 6.6.1. It is reasonable therefore for this application to consider Pattern A a *small* distortion, despite its magnitude. In summary, the descriptors do not seem to be useful to quantify the distortions proposed and therefore these cannot be compared to real distortions.

Intake pressure distortions applied constantly in time were tested and used to trigger rotating stall. However, in this case the low energy flow coming from the distorted inlet sector interacts with the rotating stall, 'anchoring' it to a specific circumferential position. As such it was preferred to use impulsive distortions and maintain a 'clean' intake for the rest of the simulation to avoid any interaction with the compressor stall.

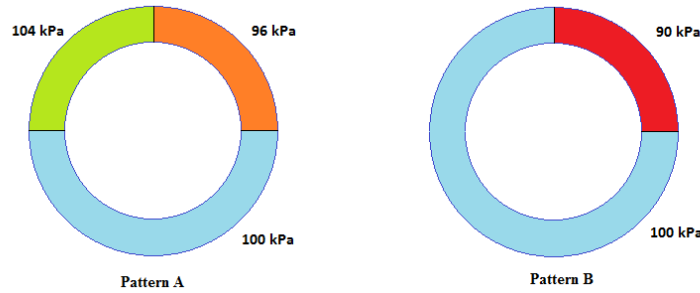


Fig. 6.20: Different patterns used at distortion inlet.

6.5.1 3-stage MIT rig

The experimental rotating stall characteristic line is formed by the data reported by Eastland (high ϕ) and Gamache (low and negative ϕ), only the 2400 RPM characteristic is fully available. All the results reported in this section are simulated at 2400 RPM.

Before modelling the rotating stall the forward flow characteristic is created. The correlations for forward, unstalled flow described by Aungier [60] are used, these are described in section 4.5. These correlations are not applicable to the IGVs because

of their shape; since the IGVs have high solidity and little camber their deviation is assumed to be zero, the ω is set to a constant low value. The forward flow results were initially found to be matching poorly with the data, possibly because the full blade geometry is not known and the correlations are tuned based on the data from NASA 2-stage rig.

The correlations are then modified using tuning factors to obtain an acceptable matching along the whole speed-line. The tuning is performed by multiplying the deviation and pressure loss coefficients obtained from the correlations with two constant factors:

$$\omega'_{SS} = \omega_{SS} \cdot C_{\omega TUNING} \quad \delta'_{SS} = \delta_{SS} \cdot C_{\delta TUNING} . \quad (6.11)$$

The forward characteristic is reproduced multiple times with various combinations of the two factors. The combination $C_{\omega TUNING} = 1$, $C_{\delta TUNING} = 0.5$ was found to give a sufficient matching along the whole unstalled characteristic, as shown in figure 6.22. The choice of these tuning factors also gives a surge margin close to what experimentally reported. The matching of the forward, unstalled flow performance is not investigated further as this is outside the focus of the present project.

Once the forward and the reverse flow characteristics are matched, the rotating stall is modelled. The first point on the rotating stall characteristic is obtained by loading a simulation of the unstalled compressor and triggering a stall. This is achieved by closing the throttle and using pattern B distortion at the inlet. The volume is set at a very low value ($V_{Pl} = 0.5 \text{ m}^3$) to ensure that a rotating stall is obtained and to speed up the simulation by having small oscillations.

The grid used has size $400 \times 10 \times 64$. The number of grid elements the meridional plane is chosen with the same method used for the 2-stage NASA rig test case, with a finer discretization in the axial direction to improve the robustness of the body force method during compressor stall. To choice of the number of circumferential element is based on what used for this application on similar codes, in particular by Longley [41]. The grid use by Longley and taken as a starting point had $131 \times 6 \times 32$; the number of element is about doubled in the radial and circumferential direction to maintain a similar aspect ratio. A grid convergence study was performed for this

Simulation parameters	
Grid size	$400 \times 10 \times 64$
Rotational speed	2400 RPM
Plenum volume	$V_{PL} = 0.5 \text{ m}^3$
Throttle variation Case 1	24% at 1 ms
Throttle variation Case 2	80% at 1 ms, 30% at 200 ms
Outlet (atmospheric) pressure	100 kPa
Inlet pressure	100 kPa
Inlet distortion	Pattern B, -10 kPa, 4 ms
Inlet temperature	288 K

Tab. 6.9: Parameters of the two rotating stall development simulations with different throttle closing schedules.

case and is presented in section 6.1. The simulation was run successfully with the compressor developing a rotating stall and finding a steady equilibrium point on the map.

The same point was reproduced with the same distortion but with two sequential throttle changes with the same final A_T as before; the trajectories of the two cases are reported in figure 6.21. The resulting point on the characteristic is the same for both simulations, demonstrating the reproducibility of the result and the absence of unexpected, non-physical hysteresis. The pressure distortion at the inlet causes a jump in the ψ measurements which can be observed in the figure, in fact both trajectories right after the start have a sudden increase in pressure rise, when distortion is applied, and then a sudden drop when it is removed. The equilibrium point is at a quite low value of ϕ and both the trajectories have temporarily negative flow during the stall development. The parameters used for the two simulations are listed in table 6.9.

From the first starting point the rest of the characteristic is obtained changing only the throttle. The results obtained on the whole characteristic are compared to the data in figure 6.22. The rotating stall characteristic was matched very well on a large range of flow coefficients. The matching of this characteristic is obtained

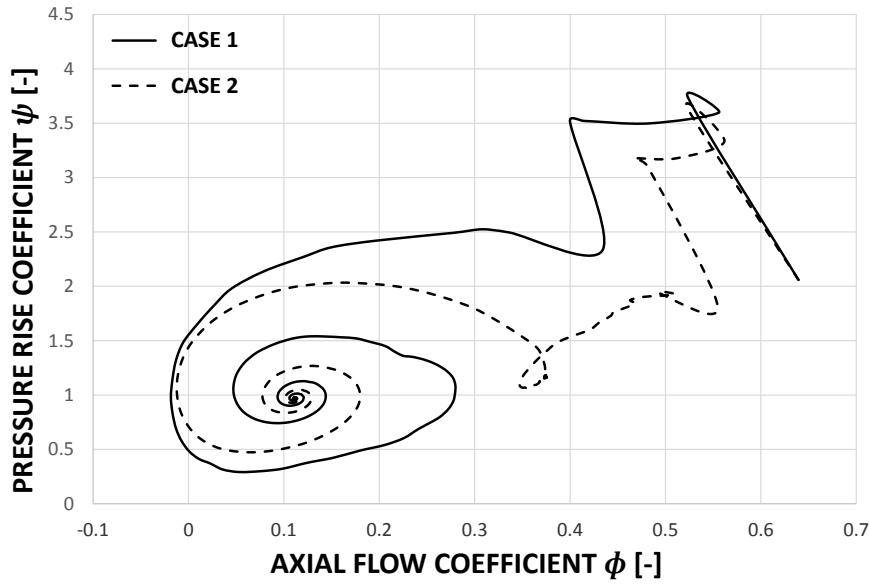


Fig. 6.21: Trajectories on the map of two rotating stall developments with different throttle closure schedules.

without any tuning, however it relies on the empirical correlations which have been calibrated to match the forward flow characteristic. Therefore the model with the current empirical correlation cannot predict the forward flow performance, but it can predict the rotating stall characteristic if the others have been matched.

Only the extremes of the characteristic were not reproduced by the model. Once this characteristic is obtained the full compressor map of the compressor can be drawn. The time required to create the whole map for this rig (forward flow, reverse flow, and rotating stall characteristic) is of 36~72 hours (not including the model set-up). The exact time depends on the accuracy required, how long the simulation are run and how many points are desired. Also, some trial and error of the boundary conditions is often necessary to obtain the desired rotating stall.

The higher ϕ values are not reached because the compressor recovers too early; a similar problem in modelling this area was reported by Longley (see figure 3.26). A possible cause is that the modelled forward flow characteristic is slightly different and may have a large influence on this part of the rotating stall characteristic, where most of the annulus is unstalled. Small steps in opening the throttle are used to reach the right extreme of the rotating stall chic but, because of computational time, it is impossible to have *infinitesimal* steps as in reality. As can be seen in

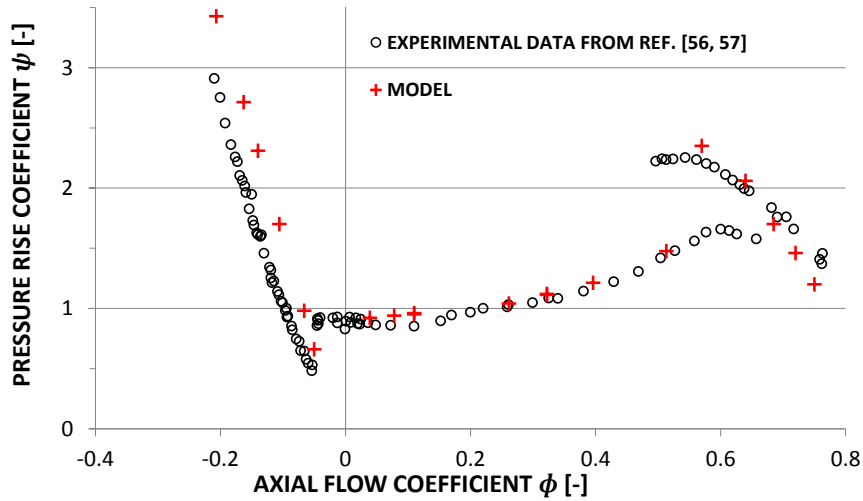


Fig. 6.22: 3-stage MIT rig complete map validation, experimental data from Eastland [56] and Gamache [57].

figure 6.23 a throttle opening causes the operating point to drop temporarily. The non-infinitesimal steps used in the throttle opening result in significant ‘drops’ in the trajectory and might cause an early recovery.

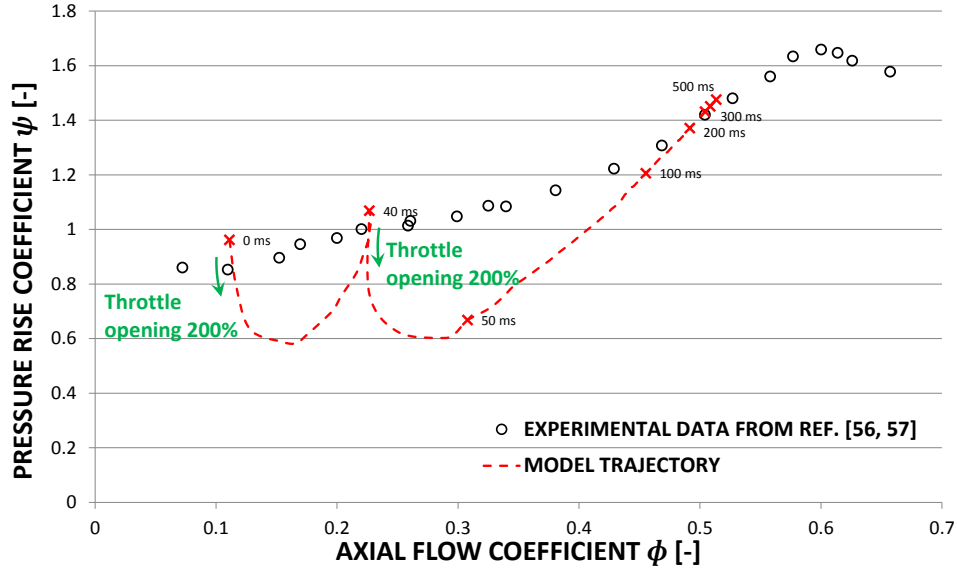


Fig. 6.23: Operating point trajectory during two consecutive throttle openings.

Lastly it is found that close to the recovery the movement of the operating point has quite a different dynamic than what has been reported in literature. Close to recovery the trajectory becomes a flat line and follows the direction of the characteristic, the operating point changes slowly and it can take multiple rotor revolutions

to reach equilibrium. This effect can be seen in figure 6.23 in which the operating point was moved by opening twice the throttle. The rotating stall is present since the beginning of the simulation and a first throttle opening of 200% is done at time 0 ms. The volume of the plenum used is extremely small, 0.1 m^3 , to accelerate the simulation convergence and there are practically no oscillations. After the first throttle opening the operating point quickly dives and rises back to the characteristic at a higher axial coefficient. 40 ms into the simulation the throttle is opened again by 200%. This time the oscillation is much larger and the operating point moves closer to the limit of the characteristic. However, as the time labels on the trajectory show, the last part of the movement takes a long time. The time required to reach equilibrium is much more than the previous ‘jump’ had required. The simulation reaches an equilibrium after about 450 ms (18 revolutions) from the throttle movement. This behaviour is possibly given by the interaction of the throttle and the rotating stall characteristic which, in that area, becomes more tangential. We can apply the linearised dynamics investigation described in section 3.1.4. As the slope characteristics of throttle and compressor are about the same, the elasticity coefficient in equation 3.4 becomes ~ 0 while the damping is non zero. As a result the equilibrium point *attracts* the operating point very weakly and causes this slow dynamic.

The compressor is also throttled to a mass-flow close to zero, down to $\phi = 0.039$ following the experimental characteristic. With a further reduction of the throttle the rotating stall is modelled down to $\phi = 0.0034$ but the value of ψ drops to 0.64. The structure of the stall cell also changes from full-span stall to a different structure where the annulus tip is reversed at every θ and a rotating stall cell of unstalled flow exists at the hub. This new structure is different from what reported experimentally and seems to occur together with the unexpected drop in pressure rise; for these reasons this last solution is considered wrong and discarded.

Also the characteristics of torque coefficient is reproduced and compared to the one reported by Gamache, figure 6.24. The definition of torque coefficient is given in section 5, Eq. 5.5

The results from the simulations have a similar trend to the experimental char-

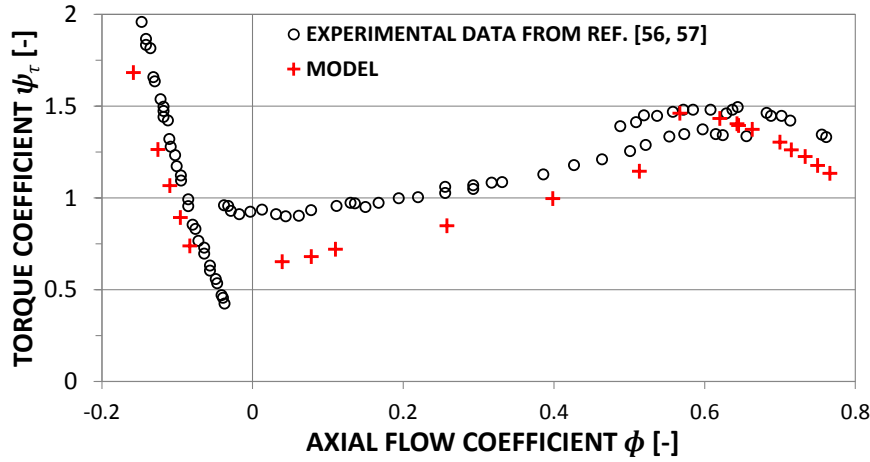


Fig. 6.24: Rotating stall torque characteristic validation on the 3-stage MIT rig full map, experimental data from Eastland [56] and Gamache [57].

acteristic but with a constant offset. Only at very low ϕ the trend of the results from the code diverges from the experimental ones. The offset appears to be roughly constant (~ -0.15) and the same for both the rotating stall and the reverse flow characteristics. It is not the only case in which this compressor torque was underestimated. As reported in the previous section both Longley in his simulations and Gamache in the post-processing of its own results found that the experimental torque was always higher than the estimated one. Unfortunately it is impossible to know whether a bias existed in the experimental measurements or the later models were wrong.

Validation of the rotating stall cell structure

Rotating stall cell size and speed are reported by Gamache and Eastland and can be used to further validate the rotating stall simulations. To simulate the experimental measurement of the stall cell speed the transient simulations were post-processed probing the axial velocity at many circumferential positions, at constant axial and radial position. The results are then plotted in a chart as in figure 6.25.

The rotating stall period t_{RSC} is calculated on every trace and averaged. From the period, the rotating stall speed can be obtained using equation 3.26. It was found during testing that stall cell speed takes a long time to settle and it is necessary to run the simulation for many revolutions after the operating point has reached

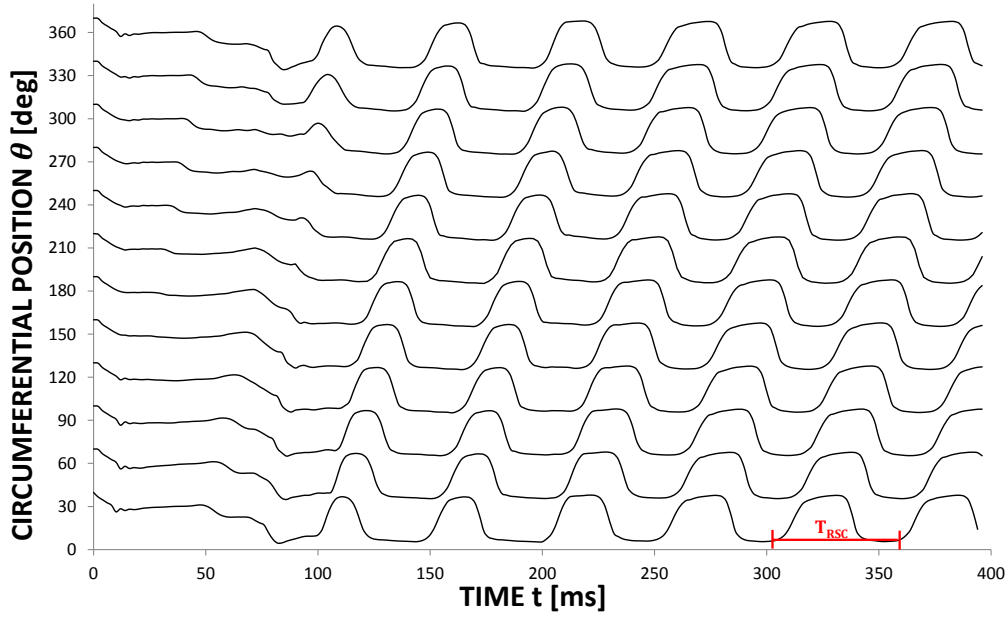


Fig. 6.25: Flow coefficient traces at different circumferential positions.

equilibrium before this measurement can be taken.

The area of the stall cell is calculated based on the axial velocities at a specific axial position. The velocities on the annular section at the specified $\bar{i}$ position, $u_{x \bar{i}, j, k}$, are analysed at a time t after the rotating stall has reached steady state. A threshold velocity is calculated as the average between maximum and minimum value.

$$u'_x = \frac{\max(u_{x \bar{i}, j, k}) + \min(u_{x \bar{i}, j, k})}{2} \quad (6.12)$$

All the elements with u_x lower than the threshold are considered part of the cell and the percentage of annulus area occupied by the stall is calculated. This definition of the threshold is somewhat arbitrary but a similar arbitrary choice is done also in the correspondent experimental measurement, see section 3.3.1. Depending on the threshold and the axial position chosen the results can be different and some ambiguity is present when comparing this to experimental data. The use of an arbitrary 50% threshold however is justified by the sharp borders of the rotating stall cell; since the change in axial velocity is large in a small space the threshold choice was found to have little effect on the final value. The axial position of the

measurement was chosen to be the same as used by Gamache, in front of the IGV.

The modelled rotating stall speed are compared to the experimental data in figure 6.26, only data from Gamache are available, for low ϕ .

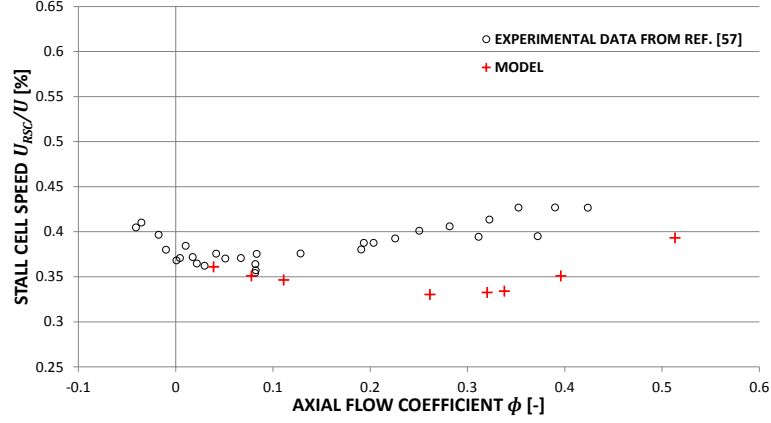


Fig. 6.26: Comparison of modelled and experimental rotating stall speed of the 3-stage MIT rig, experimental data from Gamache [57].

The modelled speed was found to be very close in range to the experimental and have the same trend of velocities rapidly accelerating close to zero mass-flow. The speed also accelerates for $\phi > 0.3$ in particular as the stall cell transitions to part-span.

The area of the rotating stall cell is compared to the experimental measurement in figure 6.27. The shape of the stall cell is also shown on the image with the area of stalled flow in black.

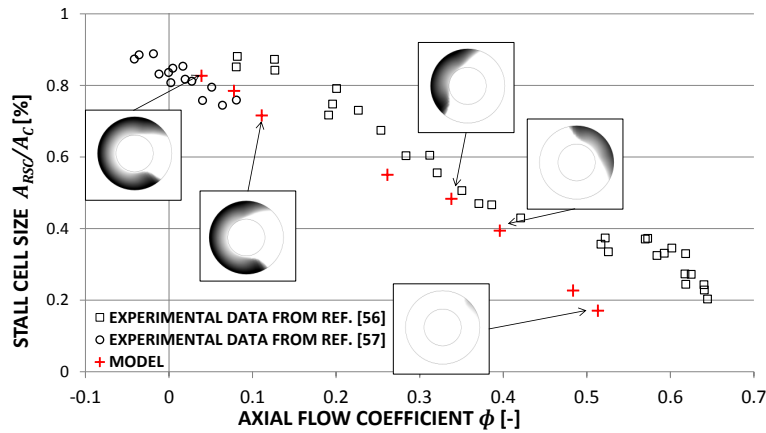


Fig. 6.27: Comparison of modelled and experimental stall cell size with examples of cell shape. Experimental data from Eastland [56] and Gamache [57].

The model simulates a single cell along the whole characteristic, in agreement with the experimental finding. The cell size decreases with the axial flow coefficient and full-span cell is observed only at low flow coefficient, the transition to part-span occurs at $\phi = 0.35$.

The stall cell size post-processed from the results matches closely the measurements from Gamache but not the one from Eastland. This can be caused by the position where the measurement, real or virtual, are taken; especially at part span the size of the cell changes significantly through the IGV. It can also be explained as a difference in the way the area is calculated from the hot wire traces. This is further supported by the discrepancy between the 2 sets of measurement available.

At higher axial flow coefficients the size of the stall cell diverges, which is reasonable considering the higher recoverability of the model. The overall trend of the stall cell shape at different flow coefficient is well reproduced. The shape of the simulated cell will be discussed in detail at the end of this section.

Investigation of the stall cell structure and propagation

The purpose of this section is to take advantage of the three-dimensional model of the mature stall cell to investigate its structure and propagation mechanism. Because of the constraints in the physical measurements this would be impossible with experimental techniques.

The full, 3D cell shape can be visualised from the post-processing of the axial velocity, non-dimensionalized as axial Mach number, as in figure 6.28 where the flow at constant radius is plotted. The color scale has been adapted so that the flow is divided in forward flow (blue) and reversed flow (red); this convention will be used throughout the rest of the chapter. The rotating stall investigated in the next section (in figures 6.28, 6.29, 6.30, 6.32) is at $\phi = 0.38$ and has a part-span cell. In all the figures the axial position x has been non-dimensionalized on the overall length of the domain.

$$x' = \frac{x_i - x_1}{x_{N_x} - x_1} \quad (6.13)$$

The blade rows have been over-imposed in black, the rotors have a positive

rotation (anti-clockwise looking at the compressor from the front). In this section and the following the cell borders will be called *propagating edge* and *receding edge*, the first is the border where the flow reverses (lower border in the figures) and the latter is the one where the flow unstalls (upper border of the stall cell in figures).

In figure 6.28 it can be seen that the stall cell has a constant width inside the 3 stages but it reduces rapidly through the IGV and upstream. The area of reversed flow extends for a few centimetres upstream of the IGV but the flow in front of the stall cell is influenced all the way to the domain inlet. The shape of the area of reversed flow is more constant downstream of the compressor and reaches the outlet with almost constant size.

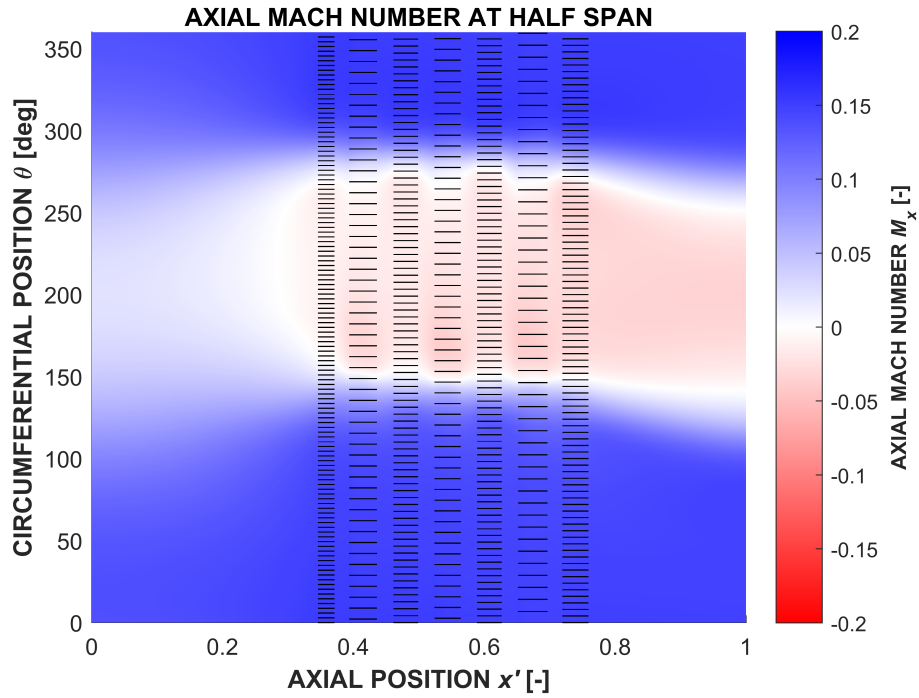


Fig. 6.28: Axial Mach number at half span (radius $r=0.285$ m) during rotating stall.

The pressure field during the rotating stall is shown in figure 6.29. The pressure has been non-dimensionalized as a static to static pressure rise coefficient

$$\psi_{S-S} = \frac{P - P_{REF}}{\frac{1}{2}\rho\bar{U}^2} \quad (6.14)$$

where the reference pressure P_{REF} is the inlet total pressure 101325 Pa

The static pressure rise is normal in the unstalled area, increasing as expected

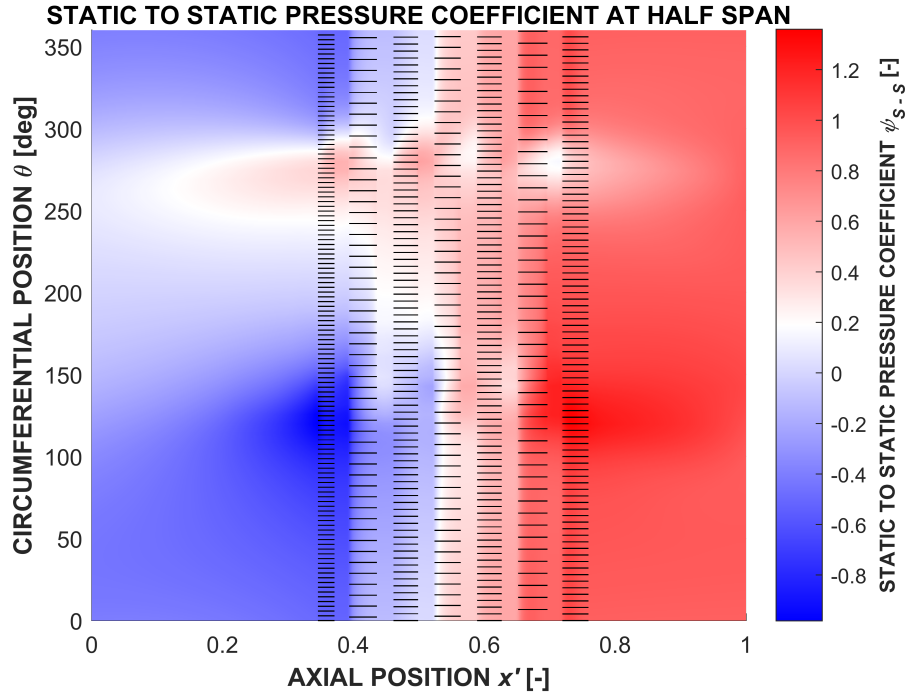


Fig. 6.29: Static to static pressure rise coefficient at half span (radius $r=0.285$ m) during rotating stall.

across the stages. In the stalled region instead, the reverse flow has a slightly higher static pressure, but the same gradient exists. In fact, the flow loses pressure while crossing the stages. Two particular features can be observed at the border of the stall cell. At the propagating edge of the cell ($\theta = 130^\circ$) there is a peak of low pressure in front of the IGV and a peak of high pressure behind the last stator. At this circumferential position the higher gradient across the compressor forces the flow reversal and therefore the propagation of the cell in the direction opposite to rotor movement. At the cell receding edge instead there are opposite peaks of pressure at compressor inlet and outlet which drive the fast unstalling of the flow.

The direction of the flow across the stages during rotating stall is investigated and reported in figure 6.30. The circumferential position scale is changed from degrees to metres to maintain the proportionality, however the image has the same dimension of the previous and cover a 360° annulus. The stall cell structure is compared to the *active stall cell* theory proposed by Day and Cumpsty [20] based on their experimental observation, figure 6.31. To be coherent with the current representation the image has been rotate of 90° and mirrored.

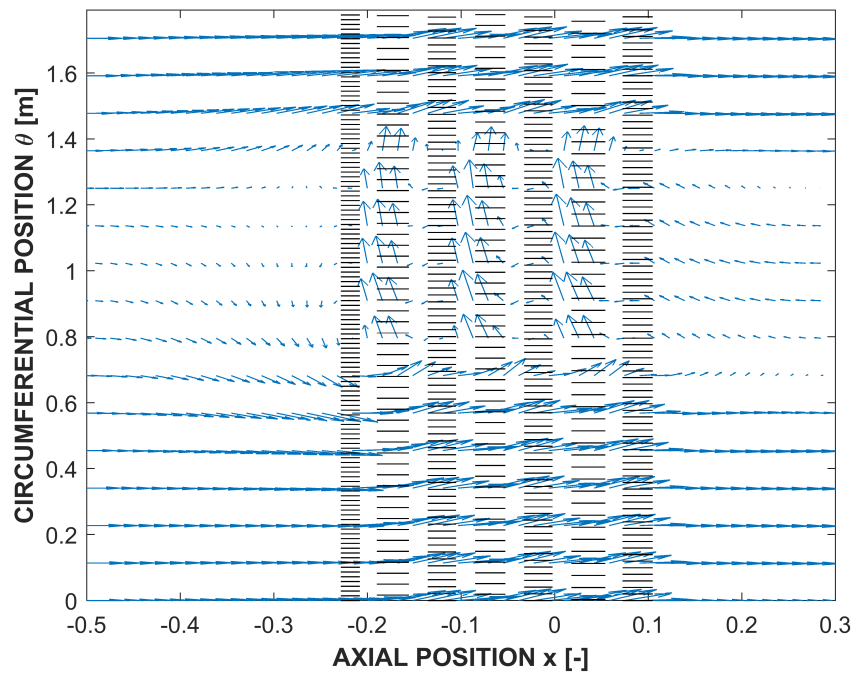


Fig. 6.30: Flow direction projected at half span (radius $r=0.285$ m) during rotating stall.

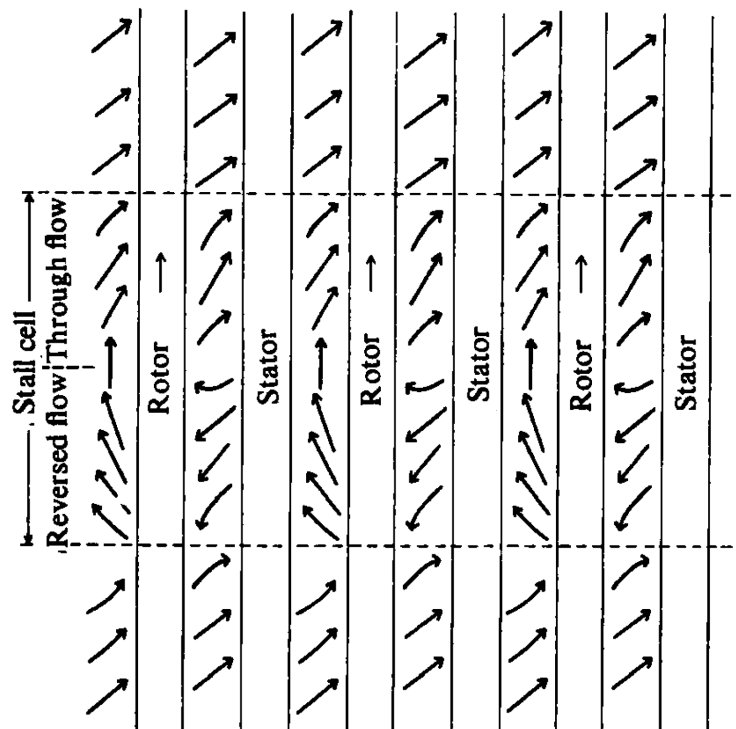


Fig. 6.31: Description of the flow direction across the stages in a stall cell as reported by Day and Cumpsty [20].

The high energy flow with a large swirl velocity in front of the rotors observed experimentally by Day and Cumpsty is reproduced in the model. The active stall cell theory divides the stall cell in a part which has a positive but low axial velocity and a part with reverse flow. The size of these two parts changes depending on the compressor geometry. The cell structure in the two figures is in general agreement, although the simulated cell appears to be formed mostly by the part which is labelled as *reversed flow* in the theoretic model. As said before the flow which exits the stall cell from the cell's receding edge is immediately re-accelerated and unstalls. However in figure 6.30 we can observe that right at the cell receding edge the flow in front of the rotors, with a high u_θ , has a small positive axial component. The flow in the stators instead has very small negative axial component. This narrow region close to receding edge seems to correspond to the *through flow* part of the cell in figure 6.31.

The flow structure in front of the IGV is not described in the active stall cell theory. From figure 6.30 it can be observed that the incoming flow redistributes around the reverse passage, similarly to what was proposed by Emmons [21]. However, since this is true only for the first row of IGV, his theory that this is the mechanism driving the stall cell propagation does not seem to be valid.

The mechanism which causes the propagation appears to be the inertia of the flow inside rotor and stator as described by Day and Cumpsty. At the cell propagating edge, in the space between the IGV and the first rotor is observed the peak of low pressure which causes the flow reversal. By observing the flow field in figure 6.30 we can see that the flow in the IGV passage immediately inside of the cell has a low axial velocity. The first rotor's motion continuously brings unstalled rotor passages with high velocity flow right behind this IGV passage. The space in-between these two rows, at the cell propagating edge, has instantaneously a low incoming flow from the IGV and a high exiting flow through the rotor. This deficit of mass-flow generates the reduction in density and pressure observed. The same mechanism is repeated at the following stages along the cell propagating edge, reversing the flow in the whole compressor. An opposite structure can be observed at the cell receding edge creating the local reduction in pressure gradient across the compressor. In this

border the rotors bring the reversed flow (from the stall cell) behind high speed flows from the stators (unstalled); the packing of the flow cause the peak of pressure which force the flow from the rotors to re-accelerate and unstall.

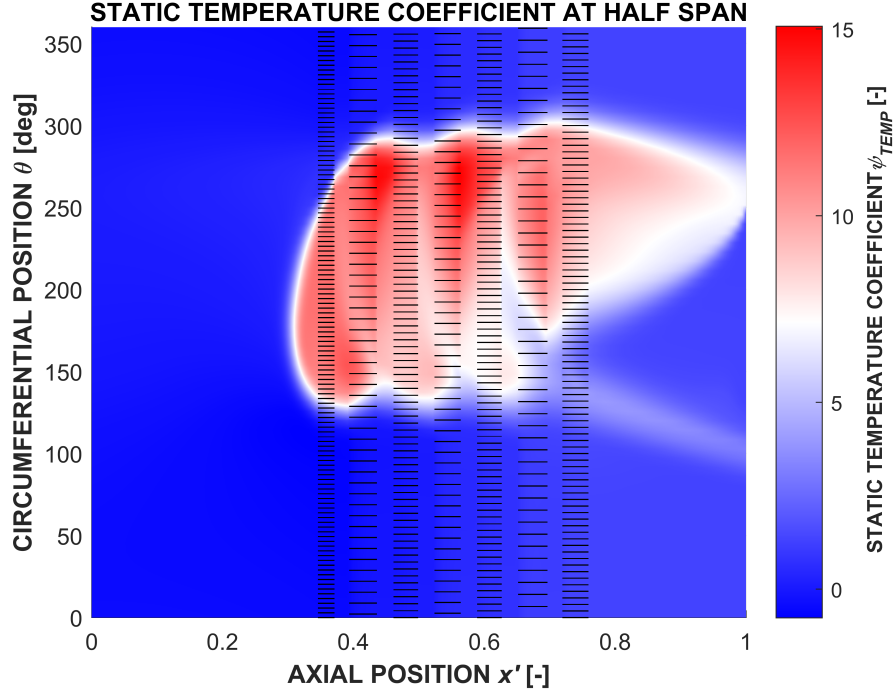


Fig. 6.32: Temperature rise coefficient at half span (radius $r=0.285$ m) during rotating stall.

The temperature profile during rotating stall was also investigated. The static temperature at half span can be observed in figure 6.32. The temperature is non-dimensionalized as a static temperature rise coefficient ψ_{TEMP} , defined as

$$\psi_{TEMP} = \frac{T - T_{REF}}{\frac{1}{2} \frac{U^2}{C_p}} \quad (6.15)$$

where T_{REF} is the inlet temperature 288 K.

The temperature inside the stall cell is higher than that of the unstalled flow and in general higher than what generated by normal operation. Cold air enters the stall cell from the last stator (the temperature of the air aspirated from the plenum is $\sim 291K$). The reversed flow is heated by the rotors as it crosses the blade rows and reaches the highest temperatures at the front of the cell. Close to the receding edge where the stalled flow has a positive axial component the highest temperature

is reached. This seems to suggest that some recirculation occurs and the flow keeps being heated by the rotors.

The stall cell is in equilibrium and, as cold air is drawn in, the heated air must be continuously extracted from inside the cell. By observing the temperature at other radii it is found that the flow at the hub, which is unstalled as this is a part span stall, is heated up as it passes ‘under’ the cell and exits at high temperature ($\sim 310K$). As heat transfer is not modelled in the Euler equations the increase in temperature must be caused by convection and the hot and cold flows mixing together. Part of the heat is also extracted at mid and high radii by some hot flows exiting the cell around its propagating and receding edges.

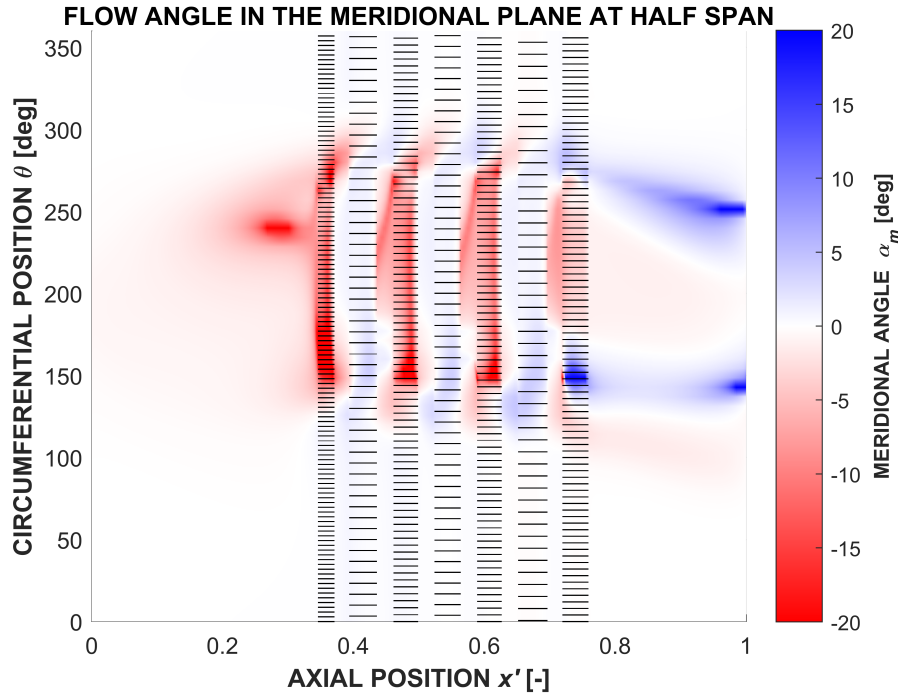


Fig. 6.33: Flow angle in the meridional plane at half span (radius $r=0.285$ m) during rotating stall.

In figure 6.33 is shown the flow angle in the meridional plane α_m during the rotating stall, defined as:

$$\alpha_m = \tan^{-1} \left(\frac{u_r}{u_x} \right). \quad (6.16)$$

At the centre of the stall cell ($180^\circ < \theta < 280^\circ$) the pattern is the same as seen during reverse flow validation, figure 6.12, with positive radial velocities (causing

negative α_m) downstream of rotors and negative velocities downstream of stators. At the borders of the cell where the flow inverts, the situation is more chaotic, with the angles going up to 90° as the flow stops and reverse; however the maximum magnitude of the radial velocity is less than 5 m/s as in these region the flow is not very energetic. Overall, in the case studied the flow is mostly axial, especially in the regions of unstalled flow, and in the reverse flow the angle does not exceed 15° . This supports the use of empirical correlations applied along pseudo-planes for the forward and reverse flow regimes.

Axial section of the stall cell

The frontal shape of the stall cell can be easily obtained by displaying the axial Mach number. In figure 6.34 the shape of the mature rotating stall can be observed in the axial Mach number at the first rotor inlet. This is at $\phi = 0.11$ and is a full-span cell. In the figure the rotors are rotating in the counter-clockwise direction, therefore the stall cell is rotating clock-wise relative to the rotor (but in a video the cell would appears rotating counter-clockwise, slower than the rotor).

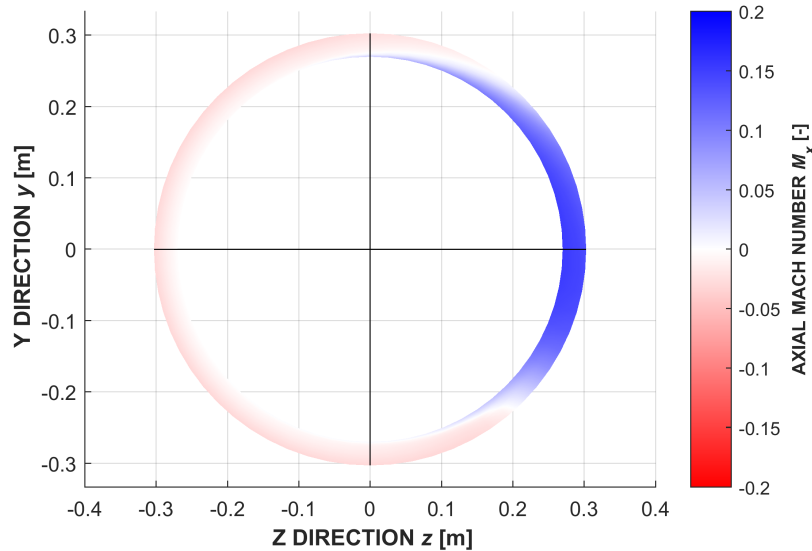


Fig. 6.34: Axial Mach number at IGV inlet during rotating stall. $\phi = 0.11$.

From this figure we can observe that the cell has a three-dimensional shape, however the high hub-to-tip ratio makes it difficult to distinguish the contours. The

post-processing of the cell shape has then been adjusted using a non-dimensional span, with the radius going from 1 to 2. The shape of the rotating stall cell is shown in figure 6.35 at different position along the 3-stage compressor.

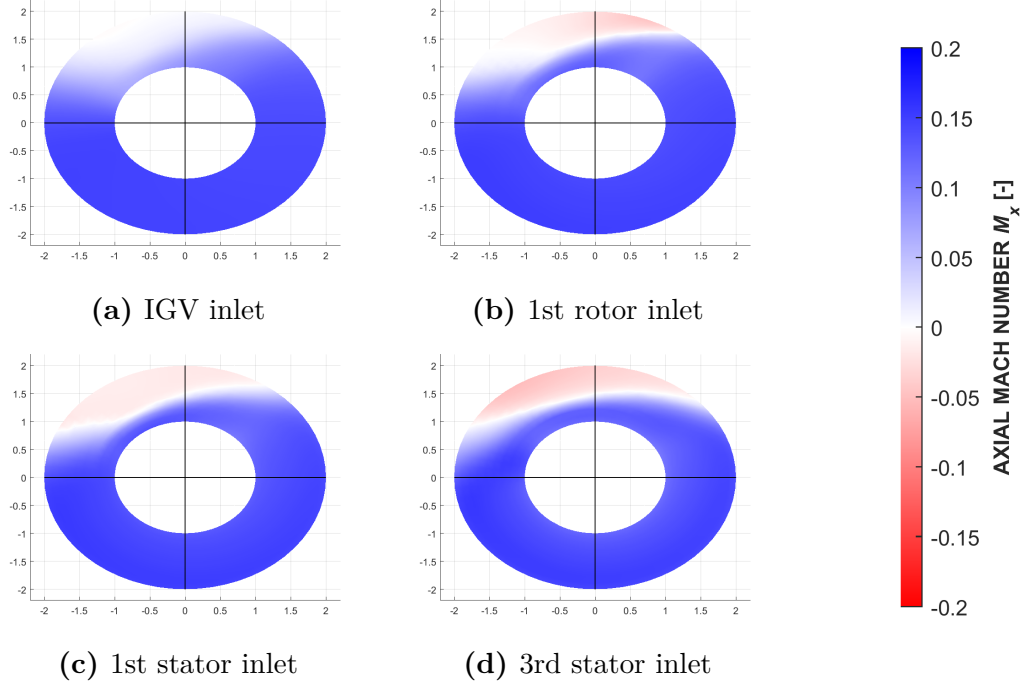


Fig. 6.35: Axial Mach number contours on the annulus section at different blade rows' inlet. Case at $\phi = 0.11$.

The stall cell has approximately the same shape across the whole compressor and as observed before there is no twisting around the x axis. The only large difference appears in the stall cell receding edge which has different shape at the IGV, in the intermediate stage and at the last stator. The positive value of ϕ for this test case ($\phi = 0.11$) is the sum of a large sector at low magnitude reverse speed and a smaller sector at high forward speed. In general, at the IGV inlet the reversed sector has a low velocity (close to 0 m/s) at the hub and a higher reversed velocity at the tip, instead the last stator has the highest reverse velocity close to the hub and lower at the tip. The intermediate stages have overall uniform flow in the reverse sector. The shapes of the trailing and leading edge of the stall cell at the first rotor are similar to that reported by Jackson (figure 3.4). The same shapes at the edges are observed also at higher ϕ . The case at $\phi = 0.38$ where the stall cell starts to be part-span is shown in figure 6.36.

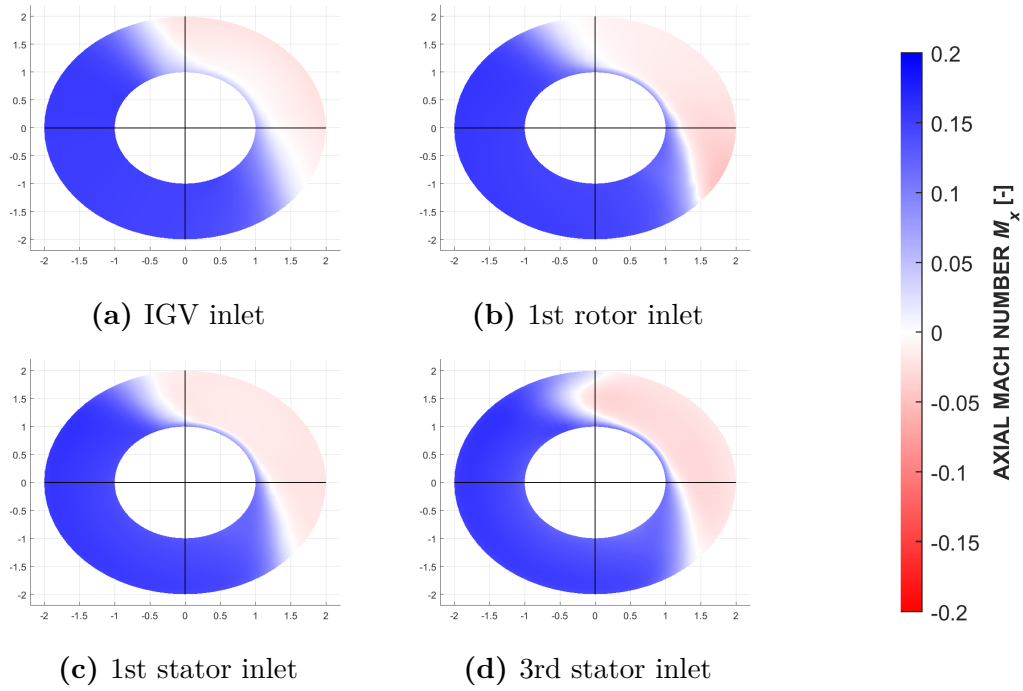


Fig. 6.36: Axial Mach number contours on the annulus section at different axial positions. Case at $\phi = 0.38$.

In this case we can see that the stall cell becomes part-span. It was decided to consider the stall part-span when the flow at the hub becomes unstalled at every axial position. Even though the flow is not reversed at the hub (or stalled), the velocity still reduces of 50% in correspondence of the cell. Therefore the definition of ‘part-span’ cell is somewhat ambiguous as in fact a reduction in flow still exists in the whole span. The 50% reduction occurs at the IGV but becomes less in the later stages. At the last stator the flow is substantially uniform at the hub. This can be observed in figure 6.36; from the scale of colors we can observe that the hub at the IGV has a minimum Mach number of 0.05, at the last stator this becomes ~ 0.15 .

The shape of the stall cell just before recovery at $\phi = 0.475$ is reported in figure 6.37.

In this last case the reversed portion of annulus covers only 50% of the span, with a stalled area of only 22% of the total. However like in the previous case at the IGV inlet the velocity is still reduced at the hub down to 50% in correspondence of the stall cell. Again, the choice of labelling this as a part span stall is somewhat arbitrary. At the last blade row instead, the stalled and unstalled regions have a

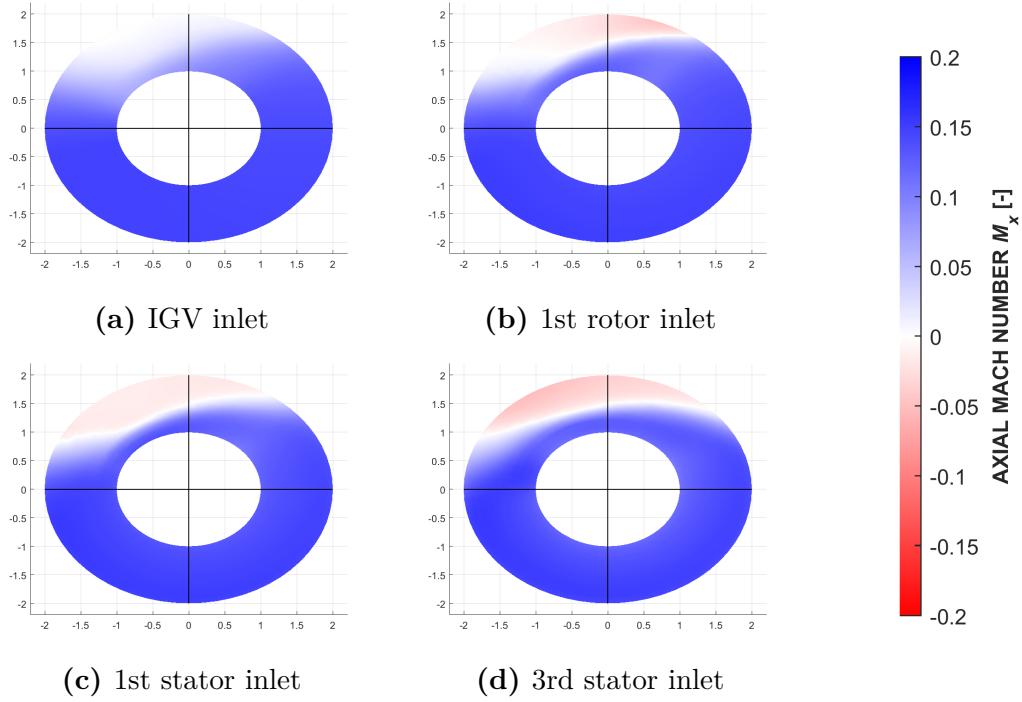


Fig. 6.37: Axial Mach number contours on the annulus section at different axial positions. Case at $\phi = 0.475$.

very sharp border, the reverse flow has a high negative velocity and the unstalled has a high positive velocity.

Eastland reported at $\phi > 0.5$ that the stall cell was still full span, as measured using a hot wire. Because of the ambiguity in definition it is impossible to determine whether the current results are in agreement or not with the experimental finding. This ambiguity can also explain the difference in stall cell size trend between the experimental data and the simulations. On the other hand, the compressor was experimentally found to suffer from rotating stall until $\phi = 0.64$ while the current tool predicts the recovery at approximately $\phi = 0.52$. It is therefore reasonable to assume that the experimentally measured cell is larger than the simulated one in this very last section of the characteristic. The stall cell extending axially through the whole compressor even at very high flow is certainly in agreement with what reported by Eastland.

On the other extreme of the characteristic the full-span stall cell was reported experimentally to cover up to $\sim 90\%$ at the maximum negative mass-flow, $\phi = -0.05$. However, as previously explained, when the simulated compressor is throttled

close to zero mass-flow the full annulus becomes reversed and a cell of unstalled flow appears at the hub, rotating in a similar fashion to normal rotating stall, figure 6.38. However this result is not considered reliable and discarded.

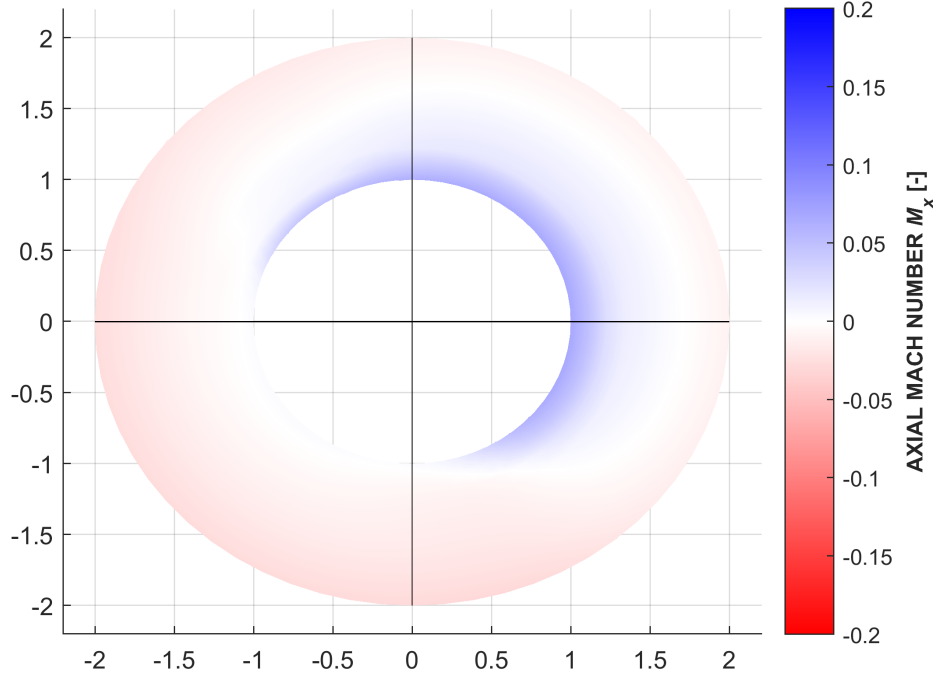


Fig. 6.38: Axial Mach number showing the simulated rotating stall shape close to zero mass-flow, case at $\phi = 0.0034$.

6.5.2 4-stage C106 rig

The same methodology used in modelling the 3-stage MIT compressor is used to model the 4-stage C106 rig and produce its compressor map.

The domain used has $600 \times 10 \times 64$ elements. The number of axial elements has been chosen to maintain the same axial element size dx as the 3-stage MIT rig case, $dx \sim 0.002 \text{ m}$. A grid convergence study has been performed following the same guideline as described in section 6.1. The simulation of a rotating stall is loaded and run for 50 ms with each grid, details of the grid tested are reported in table 6.10. The results at the end of the simulations are then compared.

Similar to what presented in the convergence study of the 3-stage MIT rig test case, the grid chosen for this case does not allow to calculate a GCI index correctly.

	Number of elements	Grid size h	Time step dt (at $t = 0$)
Grid 4	$420 \times 7 \times 48 = 188160$	$0.823 \cdot 10^{-2} m$	$2.34 \mu s$
Grid 3	$600 \times 10 \times 64 = 384000$	$0.649 \cdot 10^{-2} m$	$1.65 \mu s$
Grid 2	$660 \times 11 \times 72 = 522720$	$0.586 \cdot 10^{-2} m$	$1.48 \mu s$
Grid 1	$780 \times 13 \times 84 = 851760$	$0.498 \cdot 10^{-2} m$	$1.26 \mu s$

Tab. 6.10: Number of elements, element size, time step and computational cost of the grids used in the grid dependency study for the 4-stage C106 rig test case.

This is because the convergence occurs in grid 1,2 and 3, with the grid chosen, no. 3, being the first *step* in the convergence. It is possible however to obtain an extrapolated value of each variable X on an infinitely fine grid, and calculate an extrapolated relative error for the grid used. The variable chosen are the axial flow coefficient ϕ , the pressure rise coefficient ψ and the torque coefficient ψ_T . The results are reported in table 6.11 and the extrapolated relative error variation with grid refinement is shown in figure 6.39. While the finer grids offer a higher accuracy, grid 3 was selected as a trade-off between results accuracy and computational cost. The selected grid has a maximum relative extrapolated error of 14.5%, on the torque coefficient.

The correlations used to estimate the performances in forward flow are those developed by Rolls-Royce. The data of the full map are available only for the rotational speed of 3000 RPM [58], all the following results have been obtained at this speed. The full set of parameters used for the rotating stall simulations is reported in table 6.12, the inlet distortion was used only to obtain the first point, the other points were obtained changing the throttle. The full map is created and compared to the experimental data, as in figure 6.40.

The forward flow characteristic follows closely the experimental one but underestimates the peak ψ . The stall occurring when the throttle is reduced always starts at the hub of the last stator while Day reported the onset of stall at the first rotor tip. This could be because of either the uncertainty on the IGV geometry (which was partly guessed), an excessively conservative stall criteria at the hub geometry

Variable X	Axial flow coefficient ϕ [-]	Pressure rise coefficient ψ [-]	Torque coefficient ψ_T [-]
X_4 $420 \times 7 \times 44$	0.1264	0.5966	0.8706
$X_{3 \text{ case tested}}$ $600 \times 10 \times 64$	0.1260	0.5980	0.8990
X_2 $660 \times 11 \times 72$	0.1273	0.6108	0.9176
X_1 $780 \times 13 \times 84$	0.1279	0.6164	0.9425
X_{ex}	0.1283	0.6177	1.0509
$e_{ex \text{ case tested}}$	1.79%	3.19%	14.5 %

Tab. 6.11: Grid convergence study for the 4-stage C106 rig test case.

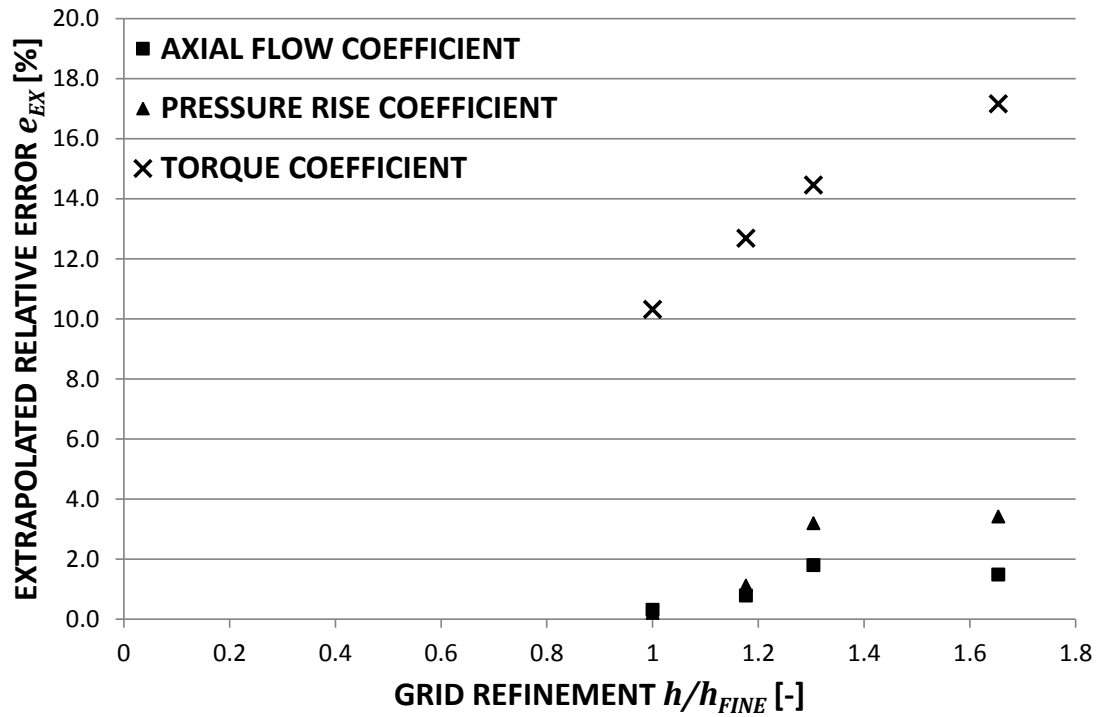


Fig. 6.39: Extrapolated errors from the grid convergence study for the 4-stage C106 rig test case.

Simulation parameters	
Grid size	$600 \times 10 \times 64$
Rotational speed	3000 RPM
Plenum volume	0.5 m^3
Outlet (atmospheric) pressure	100 kPa
Inlet pressure	100 kPa
Trigger inlet distortion	Pattern B
Inlet temperature	288 K

Tab. 6.12: Parameters of the C106 rotating stall simulations.

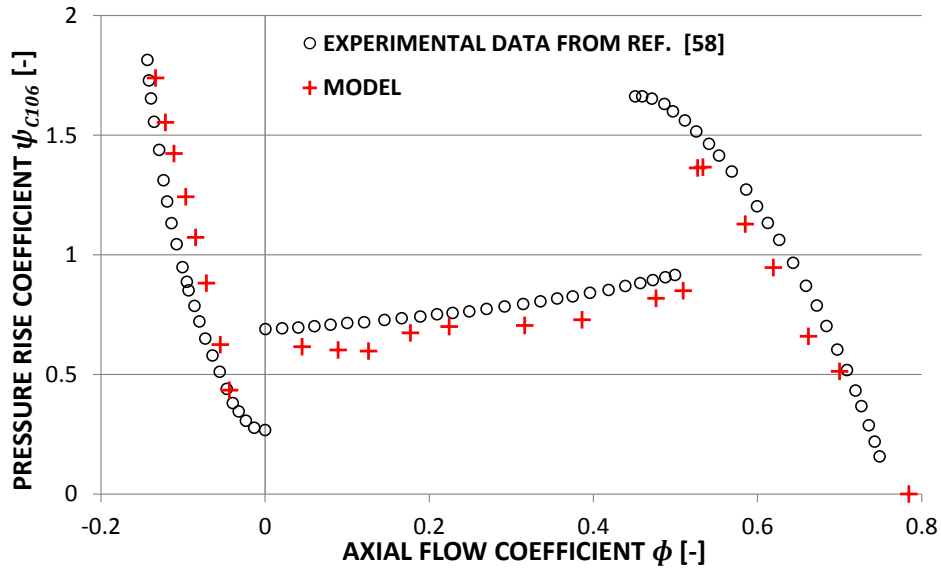


Fig. 6.40: Full compressor map validation on the 4-stage C106 rig, experimental data from Day [58].

or a poor distribution of losses and deviation along the span. Unfortunately no data are available on the profiles of loss and deviation for comparison.

The reverse flow characteristic was matched to the experimental data over most of the range, however the lowest part was not achieved as the compressor would recover earlier.

The rotating stall characteristic was reproduced with very good matching to the experimental characteristic. Although the pressure rise is underestimated, the trend is very well reproduced. Unlike the MIT rig test case the characteristic was

reproduced in the same range of ϕ showing the same recoverability at high flow coefficients.

The correct modelling of the rotating stall characteristic is obtained despite the large difference in peak value of unstalled ψ . This further supports the theory that the *mature* rotating stall mechanism does not depend on single passages stalling (the mechanism described by Emmons) but on inertia matching of the static and rotating blade rows. The blade passages do not stall because of the inlet incidence angle but are forcefully (and quickly) reversed by the pressure gradient applied. Therefore the blade passages with unstalled flow are operating at low values of ψ , far from the peak value.

The flow-field structure was not reported by Day, and validation is not possible. However the speed and size of the stall cell have been measured along the characteristic and are shown in figures 6.41 and 6.42. The flow was probed between the IGW and the 1st rotor. The tool simulated a single-cell stall in all operation conditions, in accordance with Day's observations.

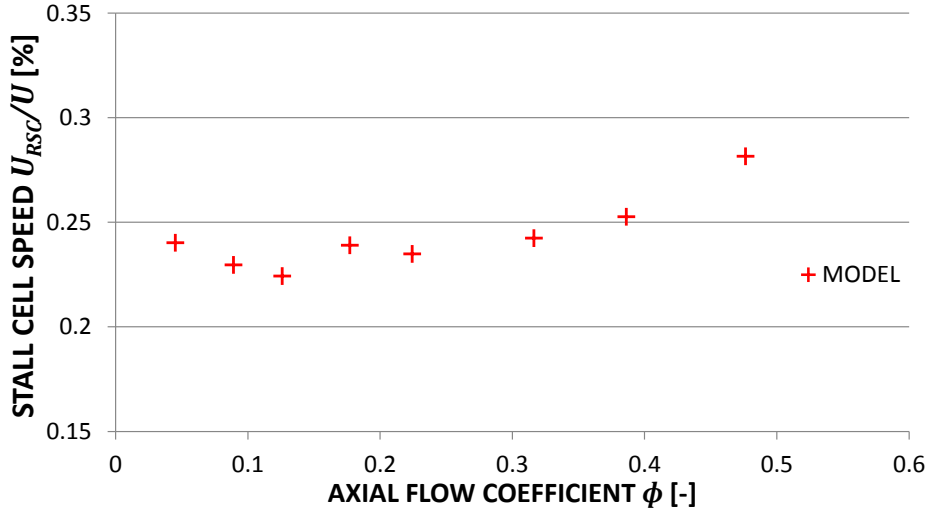


Fig. 6.41: Rotating stall cell speed modelled on the 4-stage C106 rig.

The trends of speed and size are the same as in the previous test-case, although the average cell speed is lower. The trend of the cell size along the characteristic is linear with no discontinuity, the stall cell changes from full span to half span at $\phi \sim 0.55$. At high ϕ , where the cell transitions to half span, it remains in the tip part of the annulus and across all the four stages, similar to what was observed in the

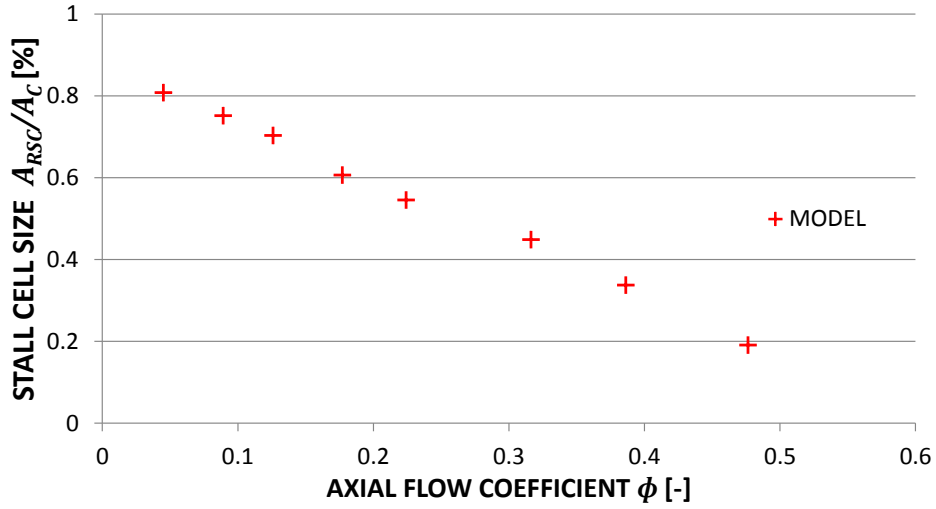


Fig. 6.42: Rotating stall cell size modelled on the 4-stage C106 rig .

3-stage MIT compressor. The cell speed increases for part-span stall and for almost zero flow coefficient. The rotating stall characteristic and the cell speed values have a small step change at $\phi \sim 0.15$, however no macroscopic change in the structure was observed.

6.6 Surge simulation

Simulations are run on both the 3-stage MIT and 4-stage C106 rigs. In this section mainly *transient* simulations are reported, where the focus is on the dynamic performance of the compressor. Depending on the simulation parameters the initial stall trigger can develop into either a surge or a rotating stall. Surge simulations can be run three-dimensional or with axi-symmetry assumption; in the following sections 6.6.1 and 6.6.2 only 3D simulations are used. A discussion on the use of 2D simulations is presented in section 6.6.3

6.6.1 3-stage MIT rig

No experimental data is available on the surge behaviour of this compressor. Gamache reported a series of surge simulations performed using a zero-dimensional model he developed. These results however are not validated and hold little value for the present work. He proposed a surge frequency estimation based on the Helmholtz fre-

Simulation parameters	
Grid size	$400 \times 10 \times 64$
Rotational speed	2400 RPM
Plenum volume	Variable $10m^3 > V_{PL} > 1m^3$
Throttle variation	85%
Outlet (atmospheric) pressure	100 kPa
Inlet pressure	100 kPa
Inlet distortion	Pattern A, ± 4 kPa, 4 ms
Inlet temperature	288 K

Tab. 6.13: Parameters of the 3-stage MIT rig surge simulations with variable plenum volume.

quency of the system but, based on what reported by Day [58], this is not considered reliable and discarded.

Response to variable volumes

The Gamache model is used for a parametric study of the surge simulation, to verify the response to a variable volume. The objective of the simulation was to verify the existence of a critical value of the volume which will change the result from a surge to a rotating stall. Every other parameter which could influence the stall dynamic is kept constant, so that the variable V_{PL} have the same effect as a variable Greitzer parameter B . An asymmetric perturbation is necessary to ensure that the compressor can develop a rotating stall if the conditions are correct. The perturbation with Pattern A (described in Section 6.5, Figure 6.20) was used as it introduces only a background asymmetry similar to what would naturally exist in a real compressor. The stall trigger is a throttle closure to mimic the operation in a test facility. The simulation parameters are shown in table 6.13. The results are collected and compared in figure 6.43.

Depending on the volume the tool simulates both rotating stall and surge as outcomes; a critical value of volume can be found, $1.5 m^3 > V_{PLCRIT} > 1 m^3$. The shapes of the surge cycles follow the trend described in literature, the higher

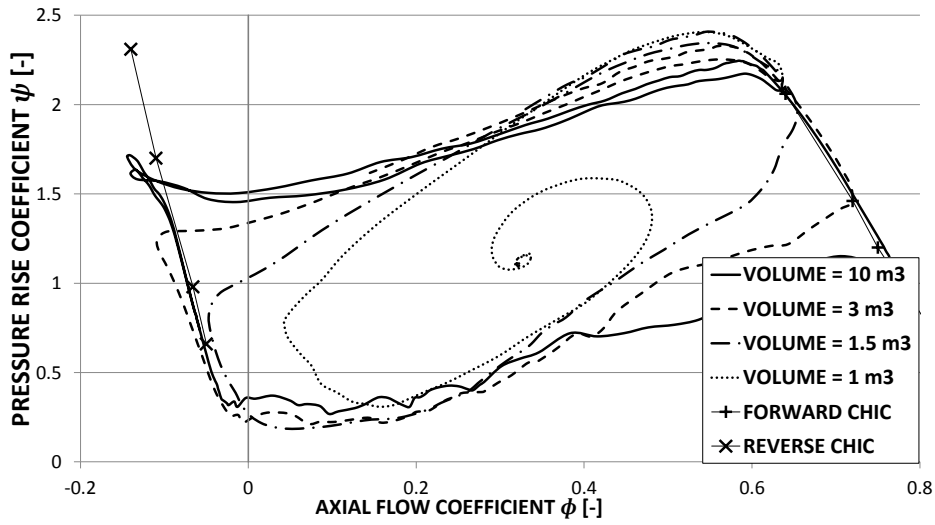


Fig. 6.43: Stall trajectories at varying plenum volume on the 3-stage MIT rig.

volume cycle reaches the reverse characteristic at the highest ψ and recovers at the lowest. The largest volume also has the minimum overshooting when leaving both the characteristics. The two surge cycle at higher volumes reach the reverse characteristic and follow it for part of the cycle, while the surge at lower volume does not fully reach it.

The simulation at the lowest volume starts with the steepest trajectory, does not reach the zero flow coefficient and during recovery part of the cycle starts the spiral trajectory typical of rotating stall. Finally it finds its equilibrium on the stall characteristic. However, when the compressor enters the spiral it is following the same trajectory as the case with $V_{PL} = 1.5 \text{ m}^3$ which instead successfully recovered. The difference in behaviour is explained by the fact that during the recovery part of the trajectory the 2 cases have completely different flow-fields, invisible in the map. The 3 cases which surged never developed a strong asymmetric flow pattern. This pattern grows during the reversing and recovery phases of the cycle; while the compressor is operating in fully reverse flow instead it reduces, and when following the unstalled forward flow characteristic it is removed completely. As the case with 1 m^3 volume never reaches the reverse flow characteristic, the asymmetry continues to grow and during the recovery a rotating stall is present. Therefore in the 1.5 m^3 and 1 m^3 volume cases the compressor has different performance during recovery: in one of them it is following the axi-symmetric characteristic and in the other one

the rotating stall characteristic.

The growth and dissipation of asymmetry during the 1.5 m^3 volume surge cycle is shown in figure 6.44. The axial velocity at 2nd rotor inlet is shown right after the stall has begun and a rotating stall is present, 6.44a, which then grows to cover the whole annulus 6.44b. The asymmetry grows during the reversal, but once it reaches the reverse flow characteristic it is dissipated and during recovery the flow is mostly axi-symmetric 6.44c. As the recovery progress the asymmetry grows again and a small stall cell is present right before the annulus unstalls completely 6.44d. Once on the forward flow characteristic the flow returns to complete axi-symmetry.

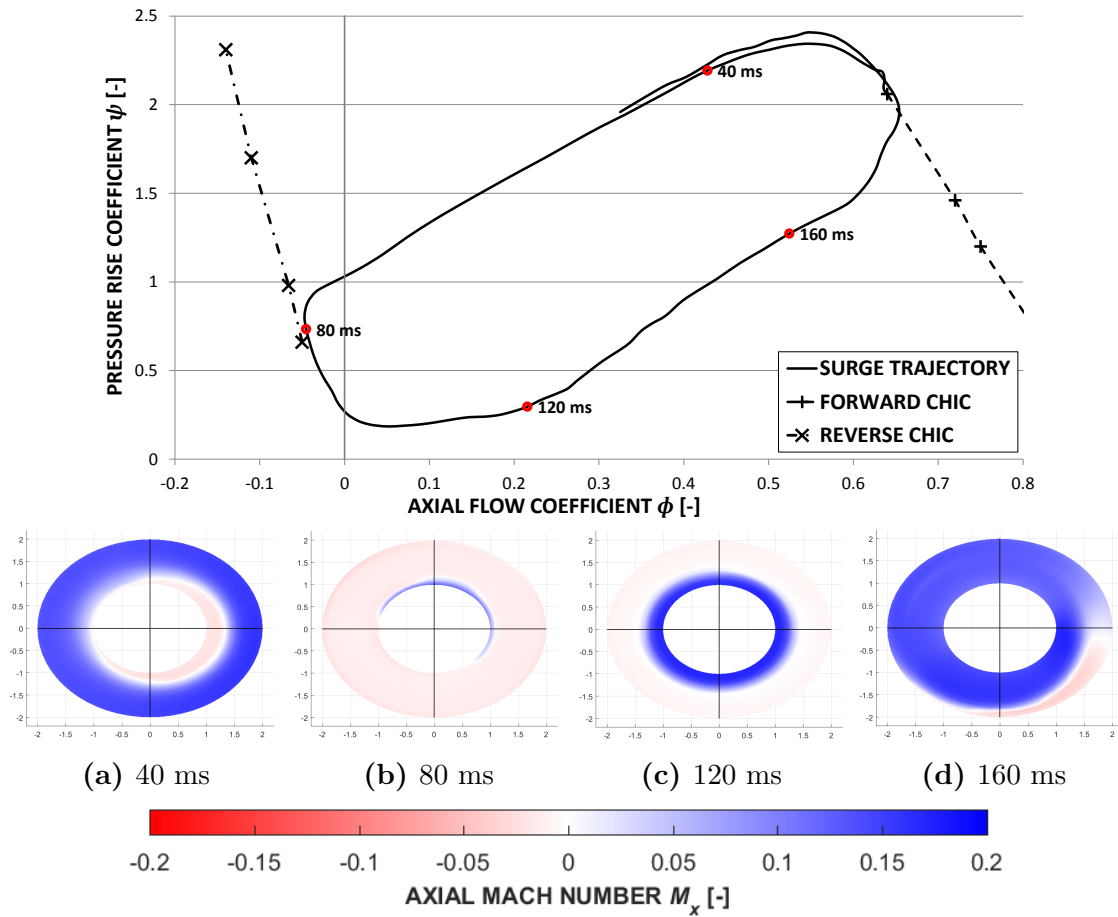


Fig. 6.44: Stall trajectories with volume of 3 m^3 , with axial Mach number at 2nd rotor inlet display the flow structure at different instants during surge.

Simulation parameters	
Grid size	$400 \times 20 \times 64$
Rotational speed	2400 RPM
Plenum volume	3 m^3
Throttle variation	85%
Outlet (atmospheric) pressure	100 kPa
Inlet pressure	100 kPa
Inlet distortion	Pattern A, 4 ms, or Pattern B, 10 ms
Inlet temperature	288 K

Tab. 6.14: Parameters of the 3-stage MIT rig surge simulations with different inlet distortions

Response to different inlet distortion

In order to assess the response of the model to different inlet distortions one of the previous simulations is re-run with the alternative type of perturbation at the inlet, Pattern B. The two patterns used have been described in Section 6.5, Figure 6.20.

Just like Pattern A, this second inlet distortion is held in place temporarily and then removed. The distortions have different magnitude and different duration in time, pattern A lasts 4 ms while pattern B lasts 10 ms. The details of these simulations are reported in table 6.14.

The resulting trajectories are shown in figure 6.45. The pattern B trajectory has been modified removing the initial part, where the trajectory on the map is erratic because of the interference of the distortion on the post-processing.

The stronger Pattern B distortion guarantees that a rotating stall cell is formed immediately and as a result the operating point spirals down onto the characteristic line. The presence of a rotating stall immediately after the distortion is removed can be observed in figure 6.46. The points of the trajectories at which the images of this figure are taken have been highlighted with $\circ$ marks in figure 6.45. Pattern A case with low distortion instead can be observed being more axi-symmetric close to full reversal. According to the previous simulation the value of V_{PL} that characterises

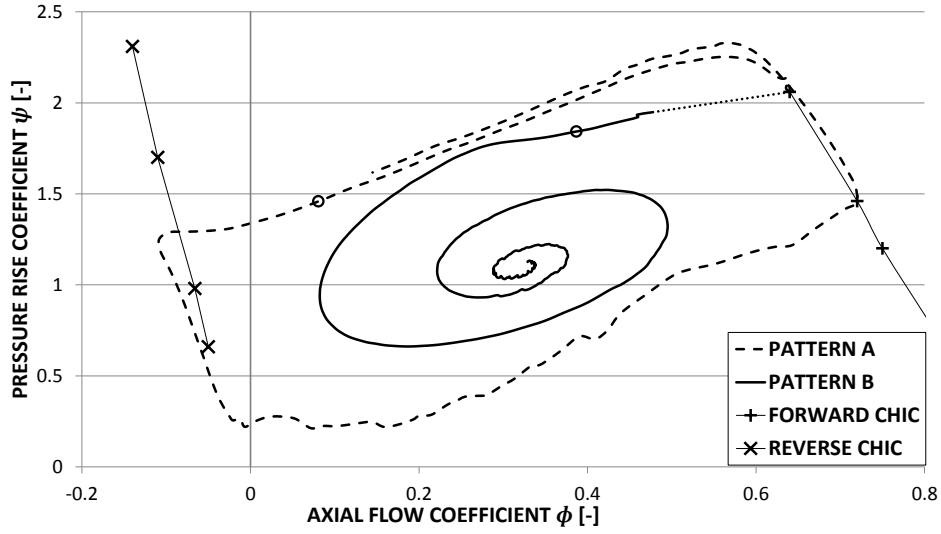


Fig. 6.45: Stall trajectories with different inlet distortions on the 3-stage MIT rig, the $\circ$ markers indicates the point where image in figure 6.46 are taken.

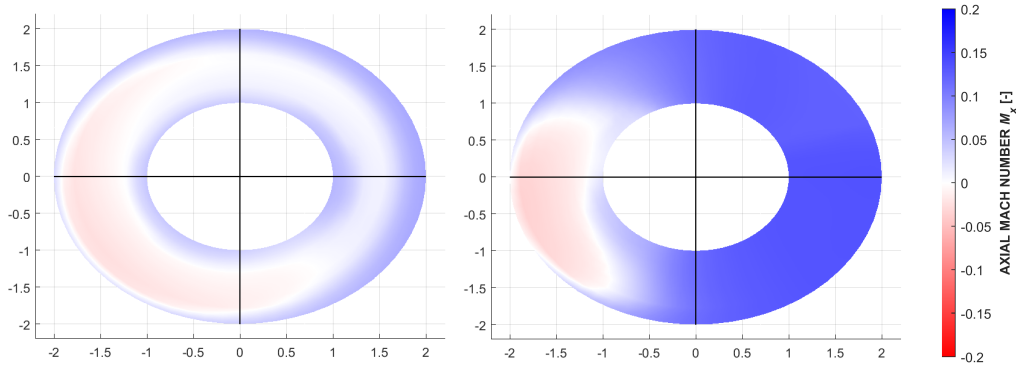


Fig. 6.46: Axial mach number at IGV inlet during surge transient. Surge triggered with pattern A shows a mostly axi-symmetric reversal (left), surge triggered with pattern B shows a rotating stall already formed from the simulation beginning (right).

the transitions from a surge cycle to a mature rotating stall is between 1 and $1.5 m^3$. The current Pattern B simulation lies far outside this boundary, yet still manages to lock in rotating stall, demonstrating the dependence on the triggering mechanism of the stall. It seems reasonable to assume that, even with the large inlet distortion, the compressor could still experience surge with a sufficiently high plenum volume. We could then say that Pattern B distortion causes the critical value of V_{Pl} to change from the previous value, $< 1.5 m^3$, to a value more than doubled, $> 3 m^3$. In this regard, the developed code proves useful as it is able to predict the outcome of two different stall triggering mechanisms.

An explanation of this particular outcome can be suggested by the linearised dynamics described in section 3.1.4. Before settling into an equilibrium both the cases exhibit mass-flow oscillation. The case which has developed a rotating stall has a much flatter compressor characteristic than the other and therefore it has smaller, more damped oscillations. For this reason the Pattern B case does not manage to return on the unstalled characteristic but remains in the central area where it keeps being attracted to the equilibrium point with progressively smaller oscillations.

The current investigation has employed two distortions with two arbitrary patterns and very different magnitudes. Since the code has proved capable of predicting the result of different distortions, it could be interesting to perform a more detailed investigation. The intensity of the inlet distortion can be quantified using one of the methods used in experimental techniques like the sectional distortion coefficient (DC60), the circumferential distortion index or the radial distortion index [66]. The results seems to suggest that there is a critical value of ‘distortion intensity’ for which the compressor stall develops into a rotating stall instead of a surge. Higher values of B parameter require higher values of inlet distortion to trigger a rotating stall and we can probably draw a profile of critical distortion magnitude for variable B and vice-versa (critical B as function of the inlet distortion magnitude).

6.6.2 4-stage C106 rig

The C106 rig was tested for surge by Day and some measurements of the cycles observed are available in his 1994 paper [58]. The surge trajectories are reported for 3 different plenum volumes $V_{PL} = 11.7; 7.2; 3.5 \text{ m}^3$, and the surge frequencies are available for seven plenum volumes from 33 to 2.5 m^3 . The same volumes are tested with the new tool for comparison.

The throttle size kept during surge in the experiments is not reported directly. In the test the throttle was moved very slowly towards the surge point, 1% reduction every 50 revolutions, and once the stall was triggered it was kept constant during the cycle. A similar technique is used during the simulation although a faster rate of throttle closure is used to save computational time. The simulated surge line is lower than the experimental one, see figure 6.40, therefore the throttle used is

larger than the real one. The different throttle is going to influence the dynamics of the surge. To test the difference caused by the throttle size, a surge cycle is simulated with the same starting point, inlet distortion and overall parameter but with different throttle closure. Two throttle closures are tested, one of 95% with a final $A_T = 266 \text{ cm}^3$ and one of 85% with final $A_T = 241 \text{ cm}^3$. The surges obtained have a different behaviour in time, as shown in figure 6.47. The smaller throttle gives a slower blow down but a faster re-pressurization than the larger throttle. Because the re-pressurization phase occupies the largest part of the cycle, the surge with smaller throttle has a smaller period and has a frequency 8% higher. The throttle closure used in the surge simulations reported in this section is 95%. It is interesting to notice that, despite the substantial effect of the throttle size on the surge dynamics, this is not included in the B parameter and is usually not reported during experimental testing. Another important effect of the lower maximum ψ is that the blow-down and re-pressurization of the tank will take less time as the amount of air stored in the volume at peak is less.

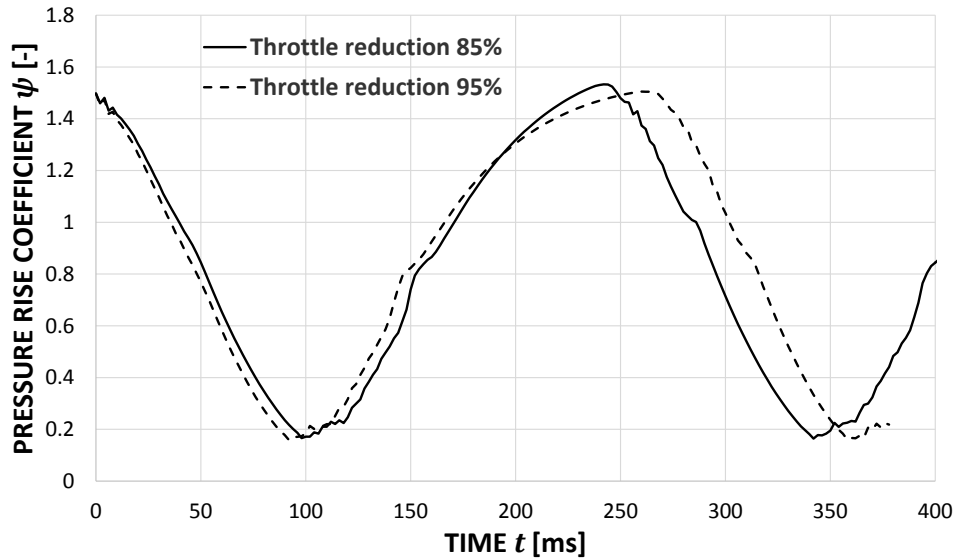


Fig. 6.47: Comparison of surge profiles obtained with different throttle closure, $V_{Pl} = 3 \text{ m}^3$.

The parameters used in the 4-stage C106 rig surge simulations are reported in table 6.15.

In figure 6.48 the surge frequencies obtained by the simulations are compared to

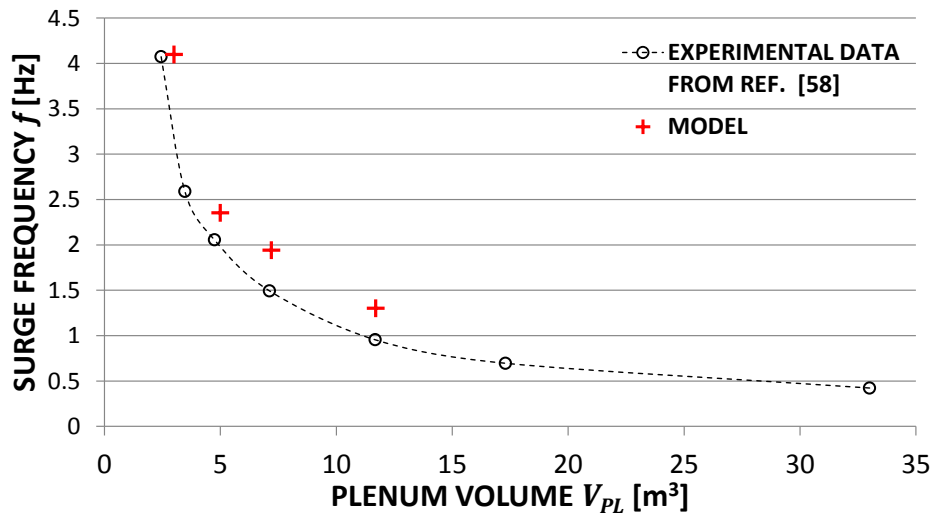
Simulation parameters

Grid size	$600 \times 10 \times 64$
Rotational speed	3000 RPM
Plenum volume	Variable: $11.7 \leq V_{Pl} \leq 3.5 \text{ m}^3$
Throttle size during cycle	266 cm^2
Outlet (atmospheric) pressure	100 kPa
Inlet pressure	100 kPa
Inlet distortion	Pattern A
Inlet temperature	288 K

Tab. 6.15: Parameters of the 4-stage C106 rig surge simulations.

the experimental data. The trend of the frequency with volume is well reproduced by the simulated cases of surge, however a constant over-estimation of the surge frequency is present. Surge cycles with volumes of 2.5 m^3 and lower result in a rotating stall.

With the chosen domain size each surge simulation, of 0.8 s of physical time, takes approximately 45 hours of computing on a single workstation with 4 cores. This time corresponds to about 1.5 to 3 surge cycles, depending on the frequency.

**Fig. 6.48:** Comparison of experimental and modelled surge frequency for different plenum volumes, experimental data from Day [58].

To investigate further the surge cycle the profile of pressure in time is also repro-

duced. In the paper from Day [58] the measurements of pressure at compressor exit and in the plenum during surge are reported. The conditions of this particular test are not described but from the timing of the single surge (and matching it with the reported frequencies) we assume that this was the experiment with $V_{Pl} = 7.2m^3$. The experimental and simulated profiles of pressure are compared in figure 6.49. The pressure rise coefficients ψ from the experimental data are calculated using equation 5.6 and considering the jump from inlet total pressure to the static pressure in the plenum (labelled *EXPERIMENTAL PLENUM*) and to the static pressure at compressor exit (labelled *EXPERIMENTAL COMPRESSOR*). The ψ from the model is calculated with the static pressure in the plenum. The pressure profile from the model was shifted in time so that the blow-down trajectories are coinciding.

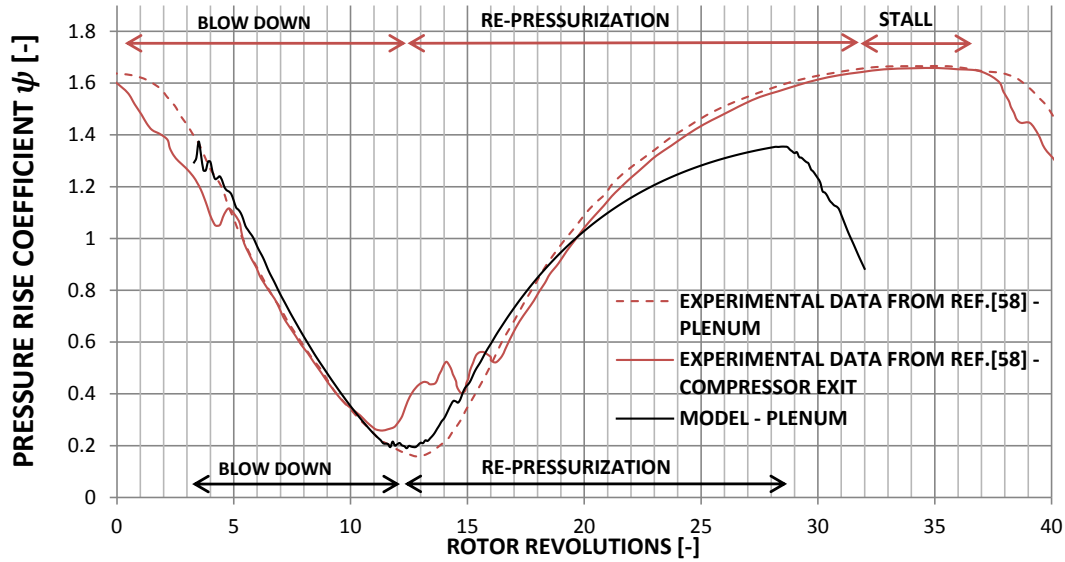


Fig. 6.49: Comparison of experimental and modelled pressure profiles during a surge cycle. Experimental data from Day [58].

From the comparison we can observe that the modelled blow down has the same rate of the experimental but lasts for less time because of the lower maximum pressure, about 3 rotor revolutions less. The modelled recovery (lower part of the cycle) occurs at a slightly higher ψ and thus it lasts less time, about 1 rotor revolutions. The re-pressurizations have a similar trend until $\sim \psi = 1$ where the modelled compressor's ψ starts becoming flat and reaches its peak at 1.38. The experimental profiles keep increasing until $\psi = 1.6$. The difference in the re-pressurization rate

at high ψ is justified by the fact that the throttle is smaller in the experiment and at $\psi = 1.2$ the real compressor is still far away from the equilibrium.

The experimental pressure profiles at compressor exit and plenum in figure 6.49 have a very specific interaction between them, in fact the difference in pressure imposes an acceleration or deceleration on the flow. At the beginning of the blow-down the lines are very distant and the pressure in the plenum, higher than at compressor exit, causes the flow to decelerate and reverse. While the operating point follows the reverse characteristic (blow down) the changes in axial velocity are small compared to change in pressure, and the two profiles are in fact coinciding. During recovery the pressure at compressor exit is much higher than in the plenum as the compressor unstalls and accelerates, until the operating point reaches the unstalled characteristic. Then during re-pressurization the plenum pressure is always slightly higher than the pressure at compressor exit, as the flow decelerates while the operating point rises along the unstalled characteristic.

The same profiles are reproduced in the surge simulation, using the ψ_{C106} and a ψ calculated using an area-averaged pressure at the last stator exit; the profiles are compared in figure 6.50. The same regions of pressure difference observed experimentally are reproduced by the simulation, as well as the plenum pressure being slightly higher than the compressor exit pressure during re-pressurization.

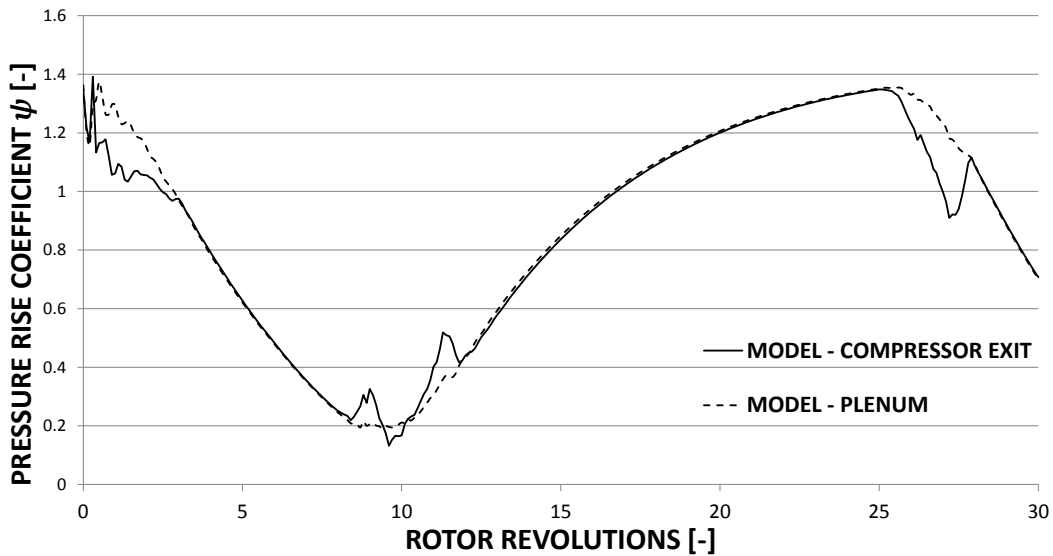


Fig. 6.50: ψ profiles measured at plenum and compressor exit during surge.

The largest difference between the experimental and modelled pressure profiles in figure 6.49 however, is caused by the stall inception itself. The modelled compressor after the first cycle reaches the maximum pressure peak and stalls immediately leaving a knee in the graph. The experimental compressor instead once it reaches the maximum pressure remains in this condition while a new rotating stall forms and grows. The pressure only decreases slightly while the stall forms before having a sudden drop once surge begins, the whole stall growth takes about 5 rotor revolutions.

At the beginning of the second cycle the simulated flow-field is absolutely axisymmetric and thus it is impossible to model a rotating stall during the stall inception. To try and reproduce as closely as possible the inception mechanism an algorithm has been developed which introduces random disturbances in the flow velocities at blade LE and TE. The disturbance is set at a very high level, 6%, to guarantee that a single virtual passage will stall before the others. However the first separation in the model occurs always at last stator hub, and not in the first rotor tip as observed experimentally. Although a single passage does stall before the others, the initial stall cell becomes axisymmetric in a small fraction of rotation and then starts to grow radially; the overall growth is so fast that the resulting profile is practically the same as in figure 6.49. The two stall inceptions are not really comparable anyway as they occur at different blade rows; it remains to be investigate how the present tool would model the stall inception if it occurred at the first rotor tip.

6.6.3 *2D and 3D surge simulations*

The simulations shown so far have been run in 3D with an asymmetry added only at the start using a distortion. The surge cycle shows transient asymmetric flow-field structures during the reversing and recovering part of the cycle, however these parts typically occupy less than 10 ~ 15% of a deep surge cycle period [58]. Regardless of the volume size and perturbation it has been observed that in the simulations the asymmetry is dissipated so much during the unstalled leg of the surge cycle that in the following cycle all the circumferential positions are numerically exactly the

same. Therefore, by running more than two cycles in succession we can observe which difference exists in the same case run with the assumption of symmetry or run in full 3D.

In this section we will use the 3-stage MIT rig case as it was tested in section 6.6.1 and vary the plenum volume, all other parameters are as in table 6.13. The case with $V_{Pl} = 3\text{ m}^3$ is shown in figure 6.51. The most substantial difference between the first 3D cycle and the second axis-symmetric cycle is in the inception and reversing part of the cycle. The distortion introduced to guarantee the asymmetry, even though it is of small intensity, makes the compressor more susceptible to stall and lowers the point at which it begins. The axis-symmetric cycle instead has a bigger overshooting. The initial part of the recovery leg is identical, as the 3D model has a mostly axis-symmetric flow (see figure 6.44). During the recovery however, as the 3D model starts to develop some asymmetry, the two trajectories begin to differ.

The overall shape of the cycle is the same and only marginal differences are present; the maximum difference in ψ is at the overshooting and is less than 5%. The difference in cycle period is also less than 5% and is caused only by the initial overshoot while the rest of the cycle occurs at substantially constant rate as can be observed in figure 6.52.

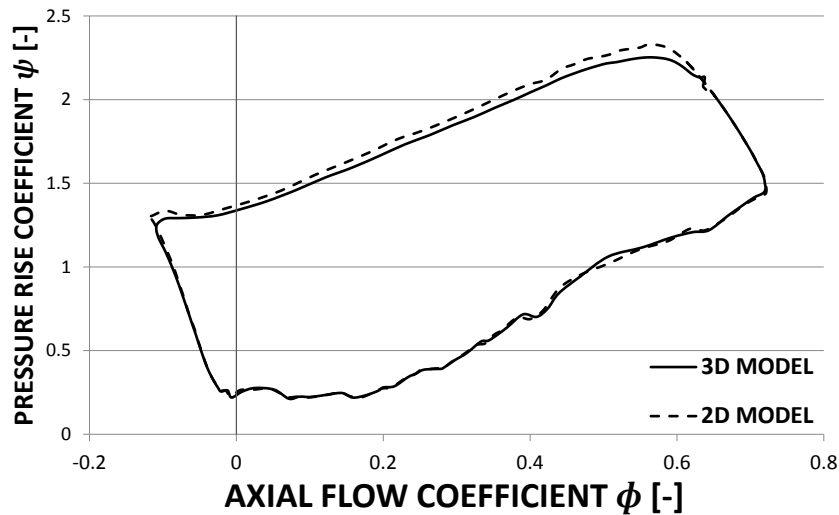


Fig. 6.51: Surge cycle of the 3-stage MIT rig, modelled with symmetry assumption and full 3D model. $V_{Pl} = 3\text{ m}^3$.

In the case with $V_{Pl} = 1.5\text{ m}^3$ the operating point spends very little time on the

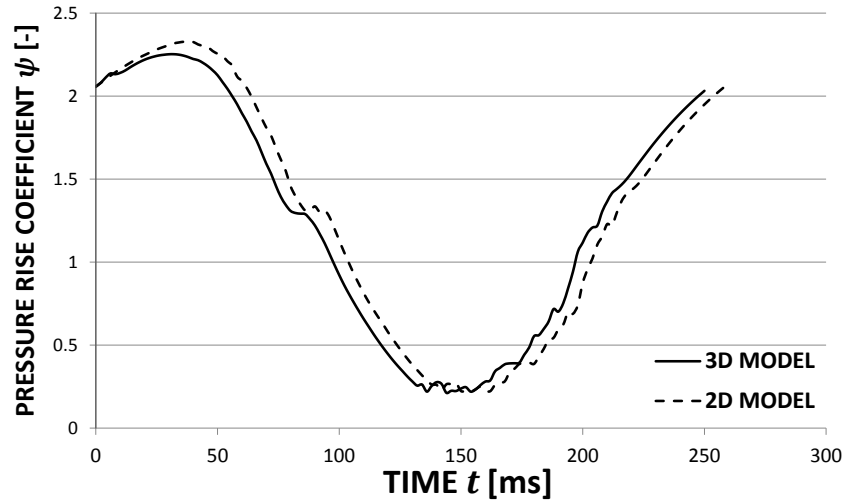


Fig. 6.52: Pressure rise profile during surge of the 3-stage MIT rig, modelled with symmetry assumption and full 3D model. $V_{Pl} = 3 \text{ m}^3$.

reverse flow characteristic and in the 3D case the asymmetry develops more. In fact the region at zero mass-flow and the recovery are quite different. In particular the 2D recovery presents a much more erratic behaviour, while the 3D one is much more smooth. Despite the differences in trajectory and the lack of asymmetry the overall profile in time remains quite similar and the period increases of less than 4%, caused again mainly by the initial overshoot.

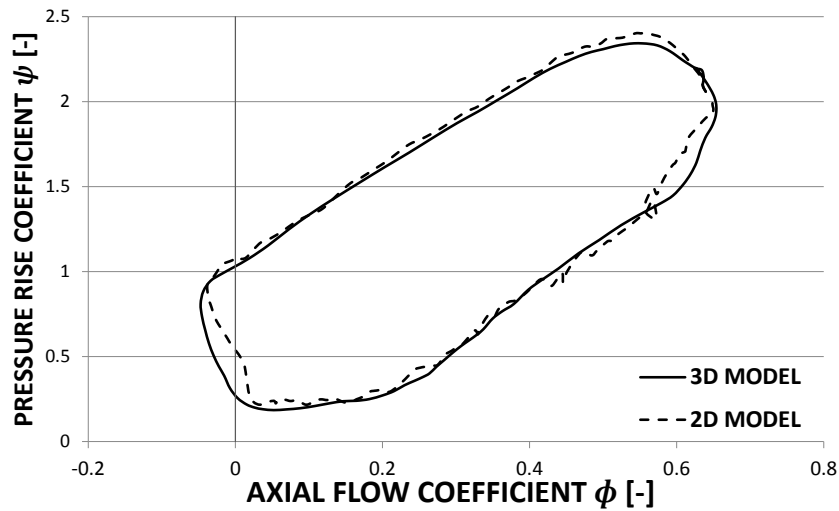


Fig. 6.53: Surge cycle of the 3-stage MIT rig, modelled with symmetry assumption and full 3D model. $V_{Pl} = 1.5 \text{ m}^3$.

When the symmetry assumption is applied to a case which should develop a rotating stall the result can be substantially different. The case of $V_{Pl} = 1 \text{ m}^3$

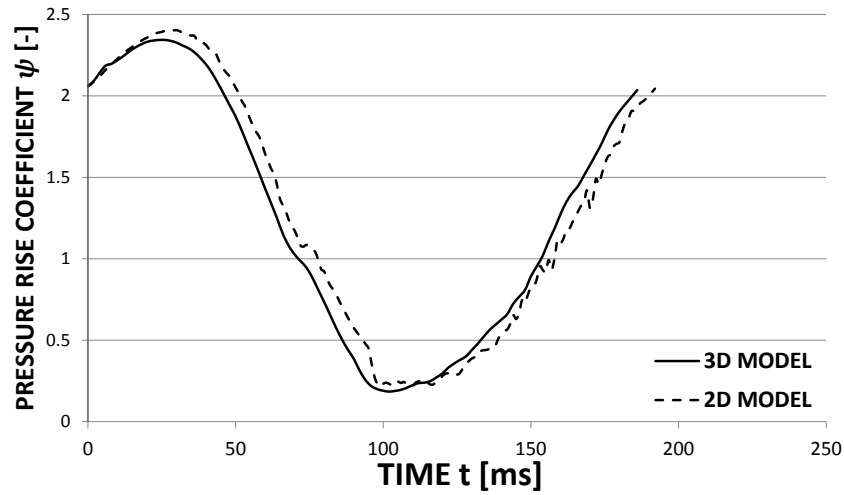


Fig. 6.54: Pressure rise profile during surge of the 3-stage MIT rig, modelled with symmetry assumption and full 3D model. $V_{Pl} = 1.5 \text{ m}^3$.

is run in 2D and the resulting trajectory is shown in figure 6.55. While this case had resulted in a rotating stall when simulated in 3D, this time the compressor suffers a classic surge. The first half of the oscillation is very similar in the two cases regardless of the presence of the perturbation. During the recovery the 3D simulation develops a rotating stall and is strongly attracted by the equilibrium point. The axisymmetric simulation instead returns to the unstalled conditions. This time the 2D simulation has been run separately and 2 cycles are drawn in the figure. Because of the short time on the forward flow characteristic some perturbation is still present when the second cycle start, and affects it. The two 2D successive cycles are then quite different. This was not observed in the previous cases, where successive 2D cycles were identical.

If an even smaller volume is tested $V_{Pl} = 0.85 \text{ m}^3$ instead the surge oscillation is too small to return on the unstalled characteristic and the operating point remains ‘trapped’ in the middle of the map where it finds an equilibrium. Because of the symmetry assumption it can not develop a rotating stall and the equilibrium is found with reverse flow and unstalled flow at different radius in the span. The flow-field in the meridional plane at the first stage is shown in figure 6.56.

The uniform flow changes radial distribution in the first rotor and a reverse flow area is formed at midspan while the rest remains unstalled. The structure is maintained constant downstream of the compressor to the domain outlet. We

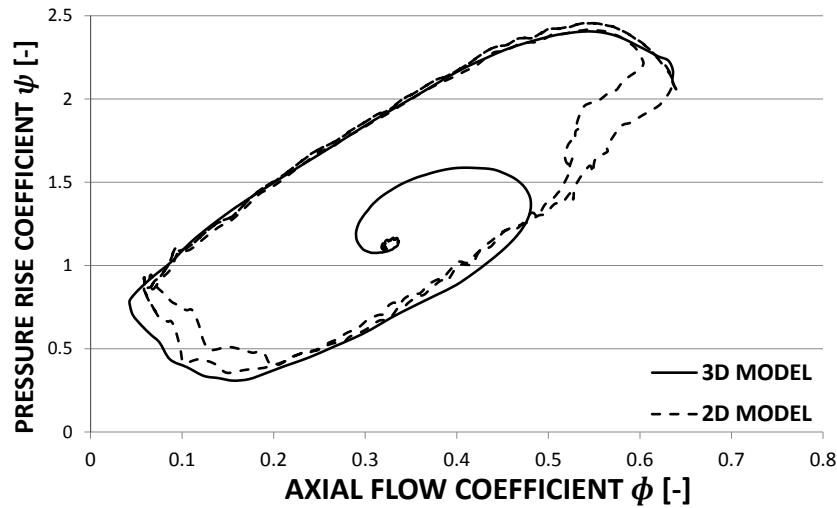


Fig. 6.55: Operating point trajectory during stall triggered with throttle closure, modelled with symmetry assumption and full 3D model. Test case: 3-stage MIT rig, $V_{Pl} = 1.0m^3$.

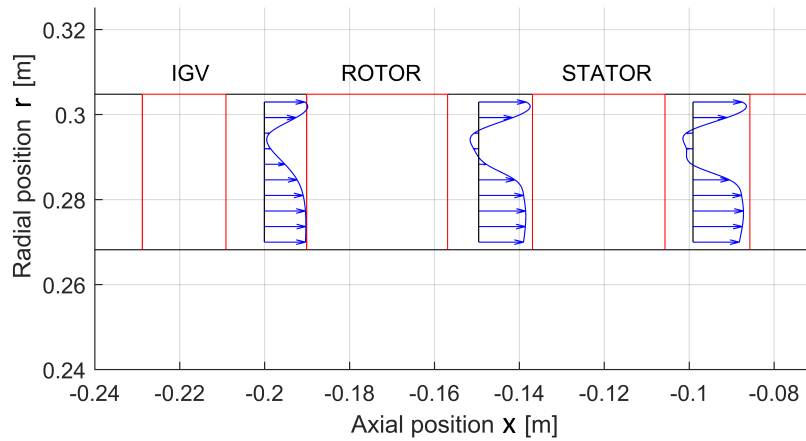


Fig. 6.56: Meridional flow structure in a failed simulation in which rotating stall was not modelled because of the symmetry assumption.

can observe that the equilibrium found has streamlines at high speed right next to streamlines of reverse flow. The compressor has a very small span, as seen in figure 6.34, and this means that the streamlines are very close to each other with a large span-wise variation in axial velocity. However, since there is no shear stress is modelled in the Euler equations this is still a reasonable solution for the tool. The absence of shear stress also allows the streamlines to continue indefinitely and not mix out even in the long downstream duct.

Concluding, when modelling a surge caused by throttle closure or a fuel spike (or

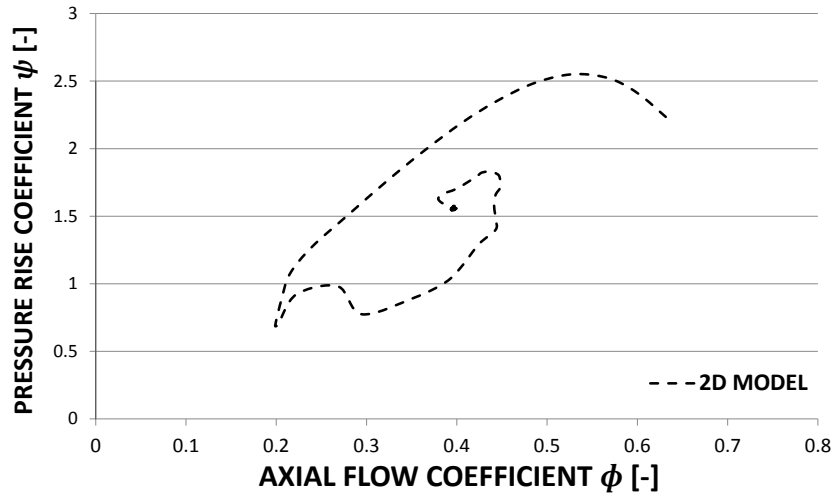


Fig. 6.57: Operating point trajectory during stall triggered with throttle closure, modelled with symmetry assumption. Test case: 3-stage MIT rig, $V_{Pl} = 0.85 \text{ m}^3$.

in general a trigger which does not introduce strong asymmetry) a 2D simulation can be used instead of a 3D. The 2D assumption should be used only in conditions where deep surge is expected and a large amount of the cycle is spent on the reverse flow characteristic. This will reduce of more than an order of magnitude the computational time and the error should be acceptable (in this test case it was $< 5\%$). When the conditions are such that the compressor is very close to a transition between surge and rotating stall ($V_{Pl} \sim V_{PlCRIT}$) then the 2D assumption might cause the tool to simulate the wrong result. More in general the 2D model should not be used whenever the asymmetry is expected to grow and having an effect on the performance.

The method used to introduce asymmetry can also be improved. A single perturbation at the beginning was used so far during surge simulations. This was not a problem as in the current study the outcome (rotating stall or surge) is decided in the first oscillation and the only case which exhibits a rotating stall is the one which does not reach the forward flow characteristic. In general however for more complex test cases there is the risk that the initial asymmetry is dissipated and does not allow the compressor to develop a rotating stall when it should. This problem can be solved introducing the asymmetry by means of continuous random perturbation of the variables in the compressor.

Chapter 7

Conclusions and future development

7.1 Summary

The objective of the current PhD project is the creation of a fast and robust tool to model three-dimensional rotating stall and surge.

In the Methodology the development of the new code is reported in detail. A three-dimensional, unsteady, compressible, cylindrical Euler solver is developed as core of the new tool. The first order Godunov scheme is chosen as its robustness, accuracy in modelling discontinuities and numerical diffusion are deemed particularly beneficial to the modelling of compressor stall transients. A body force method was created based on the formulations existing in literature, combining the desired features of each of them. The accuracy of the performance modelling depends on the empirical correlations used to model the performance of the blade rows. After a review, two sets of correlations for forward flow were identified and implemented, as well as a criterion to detect the occurrence of separation in the blade passages. A new methodology to model the performance of separated passages at low speed is developed and applied to simulate reverse flow. The capability of saving and resuming simulations is implemented in the code, and thanks to this a new methodology is developed to accelerate the creation of the compressor full map. The code is also parallelised to work on multi-core workstations.

In the Results chapter the tool is validated in successive steps using various test cases.

- A simple convergent duct is used to verify the three-dimensional Euler solver. A three-dimensional flow caused by a pressure distortion on a 90°

sector at inlet is modelled with the new tool and with ANSYS FLUENTTM; velocity and pressure profiles are successfully matched.

- Forward, unstalled flow is validated on a 2 stage compressor rig in order to test the empirical correlations and the body force method. The subsonic speed-lines at 50% and 70% of design speed are successfully reproduced on both the pressure and temperature maps. The empirical correlation were calibrated for this case and they do not match the 3-stage MIT test case, requiring additional tuning.
- The pressure and torque reverse flow characteristics are successfully matched to experimental data from the 3-stage MIT rig; the pressure reverse flow characteristic is also validated with data from the 4-stage C106 rig. Reverse flow is modelled on a 6-stage modern compressor and compared to data from CFD. Pressure and temperature characteristics are matched for low-speeds, but the simulation could not be run at transonic regimes due to lack of appropriate correlations and body forces. The overall trend is correctly reproduced at all speeds.
- The rotating stall characteristics are reproduced for the 3-stage MIT rig and the 4-stage C106 rig and matched to experimental data, obtaining the complete compressor map of both test cases. The size, shape and speed of the modelled rotating stall cell is matched to the data reported; by using the 3D model of the rotating stall the propagation mechanism and cell structure are also investigated.
- Surge is modelled on the 3-stage MIT and the effects of varying tank volume are investigated. The surge trajectory changes with volume according to theory and a critical value of V_{Pl} is found at which the outcome transitions to rotating stall. The effect of different inlet perturbations is also assessed and it is found that a stronger perturbation increases the predicted critical volume. An explanation of the results based on the flow-field structure and on surge dynamics is given.

Surge is modelled on the C106 rig and the model closely replicates the surge frequencies reported experimentally. The pressure profile during surge cycle is also successfully matched to experimental data. The main difference in the cycle shape observed is the stall inception, which is not reproduced correctly by the current tool. While during stall inception the experimental rig exhibited a growing rotating stall in the first rotor, the simulated compressor shows an axi-symmetric abrupt stall starting from the last stator.

In conclusion a new three-dimensional tool has been developed capable of producing complete compressor maps of low-speed compressors, including the reverse flow and rotating stall characteristic. The tool is able to predict the reverse flow characteristic; it can also predict the rotating stall characteristic, provided that the forward flow characteristic is correctly matched. The forward flow matching can require the use of tuning factors as the empirical correlations implemented do not have general applicability. The developed tool is capable of reproducing the full compressor map in less than 72 hours. This computational cost is unrivalled by modern 3D high-fidelity CFD codes which, when employed to model rotating stall on a full annulus, currently require weeks to model a few blade rows [48]. Three-dimensional surge simulations on a low-speed rigs with 7 blade rows can be performed with a computational cost of about 50 hours for 1 s of physical time. The surge and rotating stall results were successfully validated against two test cases, reproducing in both cases the full compressor map and realistic transient performance. All the objectives set for this PhD project have been achieved satisfactorily.

7.2 Contribution to knowledge

During this project a new attempt was made at creating a low-order method to model rotating stall and surge in low-speed multi-stage compressors. With its extensive validation on two experimental rigs, the current code already represents a step forward from most of the low order methods existing. Aside from this, the following contributions have been made to the current literature:

- The code is the first application of the physically correct Godunov solver to

three-dimensional modelling of rotating stall and surge. The features of the solver, robustness and accuracy in modelling discontinuities, are deemed particularly beneficial for this application.

- A new physic-based formulation of body forces and empirical correlations has been proposed, specifically developed to model the performance of blade rows during low speed, reverse flow and mixed flow. This new formulation proved to be general enough to be satisfactorily applied to different multi-stage compressors without extensive re-calibration necessary.

7.3 Recommendations for future work

The new tool has been successfully validated on low-speed experimental compressors, however the application to high-speed modern compressors still requires further improvements. In order to correctly simulate the behaviour of real, modern aero-engine compressors during stall the following improvements are deemed necessary:

- Improve the body force formulation to be applicable to transonic and supersonic flows. Currently the body forces become singular at $M = 1$, a new formulation should be developed.
- Improve the correlation for reverse flow deviation to include the effect of compressibility. The current assumption of constant density is used to simplify the model and thus the axial velocity is considered constant in the passage. A new correlation should be developed considering the change in axial velocity in the blade passage.
- Expand the applicability of the forward flow correlations to include transonic conditions and shock-waves inside the blade passages. Empirical correlations which predict the losses due to shock-waves exist already [69]; however it is necessary to assess their interaction with the Euler solver. In fact in transonic conditions the solver will also simulate a shock-wave, with the corresponding pressure loss. If there is a shock-wave modelled by the solver in the passage

and a shock-wave introduced by the correlation as a pressure loss then the effect will be considered twice.

- Add new functionalities to simulate the stall-prevention devices present in real engine compressor. VIGV and VSV can be introduced by implementing an algorithm which rotates the blade. However the time necessary for the VSV to change position is very long with respect to the timing of the flow dynamics, and only the effect on the steady-state characteristic is of interest. Therefore it is reasonable to avoid any dynamic change, and simply have an algorithm which modifies the inputs. Bleed valves play an important role in the performance of a stalled compressor, therefore rotating stall characteristic should be created also with the bleed valves open. They can be introduced as source terms in the wall elements, where mass-flow is removed.
- Include the possibility of having multiple shafts and therefore simulate an HPC and IPC/booster system. This will allow to investigate the interaction of two compressors during stall.
- Improve the gas model to a thermally perfect gas, with specific heat (C_V and C_P) function of the temperature T . The assumption of calorically perfect gas used in this thesis is appropriate for the low speed cases studied, where the temperature does not exceed $400 \sim 500K$. However during reversing of a six-stage high-speed compressor the temperatures can increase beyond $1000 K$, as seen in section 6.4, in this condition the new model of gas becomes necessary.
- Improve the model capability of simulating stall inception. To simulate in the most realistic way an inception caused by a throttle closure (or fuel spike) it is necessary to develop and test an algorithm which introduces random disturbances at blade LE and TE. During the present work a first attempt was done at simulating stall inception on the 4-stage C106 rig. However the attempt was flawed as the stall criteria predicted that the first separation would occur in the wrong blade row. The criterion should be better tuned so that the stall begins at the first rotor tip; it will then be possible to assess if the

current model correctly simulate the stall inception or if further improvements are necessary.

The Godunov scheme employed in the new tool has proved to be fast and robust, and the numerical diffusion it introduces makes the simulation very stable. However it is only first order accurate and the viscosity is not modelled in a way which can be controlled by the user. To overcome these problems two improvements can be considered:

- Upgrade the Godunov scheme to a second order accurate in space MUSCL scheme. To maintain the TVD propriety of the scheme a slope limiter algorithms will be necessary. The second order scheme has much less diffusivity than the one currently employed and the flow will be more similar to an inviscid flow, which is likely to make the code more unstable. It would then be necessary to model the viscosity in the equations to reintroduce its stabilizing effect.
- Introduce the viscosity in the equations. The current model, which introduces viscous effects only through the correlations, can predict flows at high speed and opposite direction extending indefinitely along the duct, a clearly non-physical result. To save on the computational time it is possible to model only the shear stress on the meridional plane as it was done by Gallimore, [45]. Introducing the viscosity along every plane does not seem necessary in the current study. However this should be re-assessed once the tool is used to simulate modern high-speed compressor.

While the introduction of a viscous term would increase the computational cost, the presence of viscosity would guarantee a faster convergence to steady state of the current model. However since the tool is intended to be used also for transient simulations, this trade-off would not always exist and the impact of the new term on the computational cost should be carefully assessed.

References

- [1] Rulexip (2011) *PAK FA T 50 compressor stall on MAKS 2011*. [Photograph] Available at: https://commons.wikimedia.org/wiki/File:PAK_FA_T_50_compressor_stall_on_MAKS_2011.jpg (Accessed: 12 December 2017).
- [2] Day, I. J. (2016) Stall, 'Surge, and 75 Years of Research', *Journal of Turbomachinery*, 138(1), pp.329-336.
- [3] MacMillan, W. L. (1974) *Development of a Modular Type Computer Program for the Calculation of Gas Turbine Off Design Performance*, PhD Thesis, Cranfield University.
- [4] Saravanamuttoo, H. I. H., Rogers, G. F. C., Cohen, H. and Straznicky, P. V. (2008) *Gas Turbine Theory*, 6th edition, Harlow, England: Pearson education.
- [5] Paduano, J. D. (2001) 'Analysis of compression system dynamic', [Lecture paper], *Active Control of Engine Dynamics*, Research and Technology Organization and Von Karman Institute, 14-18 May.
- [6] Walsh, P. P. and Fletcher, P. (1998) *Gas turbine performance*, 2nd edition, Oxford: Blackwell Publishing.
- [7] EASA (2010) *Certification Specifications for Engines CS-E*, 3 December, Available at: https://www.easa.europa.eu/system/files/dfu/CS-E_Amendment3.pdf, (Accessed: 12 December 2017).
- [8] Soria, C (2014), *Gas Turbine Shaft Over-speed / Failure Modelling; Aero/Thermodynamics Modelling and Overall Engine System Response*, PhD Thesis, Cranfield University.
- [9] Gallar, L. (2010) *Gas Turbine Shaft Over-speed / Failure Performance Mod-*

- elling; Aero/Thermodynamics Modelling and Overall Engine System Response*, PhD Thesis, Cranfield University.
- [10] Pawsey, L. (2018) *untitled*, PhD Thesis, Cranfield University.
- [11] Zoppellari, S. (2014) *Analytical Modeling of Rotating Stall and Surge*, PhD Thesis, Cranfield University.
- [12] Tan, C. S., Day, I. J., Morris, S. and Wadia, A. (2010) ‘Spike-type compressor stall inception, detection, and control’, *Annual Review of Fluid Mechanics*, 42(1), pp.275–300.
- [13] Camp, T. R. and Day, I. J. (1998) ‘A study of spike and modal stall phenomena in a low-speed axial compressor’, *Journal of Turbomachinery*, 120(3), pp.393–401.
- [14] Day, I. J. and Freeman, C. (1994) ‘The Unstable Behavior of Low and high Speed Compressor’, *International Gas Turbine and Aeroengine Congress and Exposition*, Cincinnati, Ohio, May 24–27, New York: ASME, 93-GT-26, pp.1–9.
- [15] Day, I. J. (1993) ‘Stall Inception in Axial Flow Compressors’, *Journal of Turbomachinery*, 115(1), pp.1–9.
- [16] Jackson, A. D. (1987) ‘Stall Cell Development in an Axial Compressor’, *Journal of Turbomachinery*, 109(4), pp.492–498.
- [17] Pullan, G., Young, A. M., Day, I. J., Greitzer, E. M. and Spakovszky, Z. S. (2012) ‘Origins and Structure of Spike-type Rotating Stall’, *Journal of Turbomachinery*, 137(5), pp.1–13.
- [18] Cumpsty, N. A., McDougall, N. M. and Hynes, T. P. (1990) ‘Stall inception in axial compressors’, *Journal of Turbomachinery*, 112(1), pp.116–125.
- [19] Dodds, J. and Vahdati, M. (2015) ‘Rotating Stall Observations in a High Speed Compressor—Part I: Experimental Study’, *Journal of Turbomachinery*, 137(5). Available at: doi/10.1115/1.4028557 .

-
- [20] Day, I. J. and Cumpsty, N. A. (1978) ‘The measurement and interpretation of flow within rotating stall cells in axial compressors’, *Journal of Mechanical Engineering Science*, 20(2), pp.101-114.
- [21] Pearson, C. E., Emmons, H. W. and Grant, H. P. (1955) ‘Compressor surge and stall propagation’, *Transaction of ASME*, 77(1), pp.329-336.
- [22] Cumpsty, N. A. and Greitzer, E. M (1982) ‘A Simple Model for Compressor Stall Cell Propagation’, *Journal of Engineering for Power*, 104(1), pp.170-176.
- [23] Longley, J. P. (1994) ‘A Review of NonSteady Flow Models for Compressor Stability’, *Journal of Turbomachinery*, 116(2), pp.202-215.
- [24] Aviation Safety Network (1993) *Accident description* available at: <https://aviation-safety.net/database/record.php?id=19911227-0> (retrieved 28/11/2017).
- [25] Moore, F. (1984) ‘A Theory of Rotating Stall of Multistage Axial Compressors: Part I—Small Disturbances’, *ASME J. Eng. Gas Turbines Power*, 106(2), pp.313–320.
- [26] Moore, F. (1984) ‘A Theory of Rotating Stall of Multistage Axial Compressors: Part II—Finite Disturbances’, *ASME J. Eng. Gas Turbines Power*, 106(2), pp.321-326.
- [27] Moore, F., (1984) ‘A Theory of Rotating Stall of Multistage Axial Compressors: Part III—Limit Cycles’, *ASME J. Eng. Gas Turbines Power*, 106(2), pp.327–334.
- [28] Longley, J. P. and hynes, T., (1990) ‘Stability of Flow Through Multi-stage Axial Compressors’, *Journal of Turbomachinery*, 112(1), pp.126-132.
- [29] Hynes, T. P. and Greitzer, E. M. (1987) ‘A Method for Assessing Effects of Circumferential Flow Distortion on Compressor Stability’, *Journal of Turbomachinery*, 109(3), pp.371-379.

-
- [30] Horlock, J. H. and Denton, J. D. (2005) 'A Review of Some Early Design Practice Using Computational Fluid Dynamics and a Current Perspective', *Journal of Turbomachinery*, 127(1), pp.5-13.
- [31] Mikhailov, A. E., Mikhailova, A. B. and Akhmedzyanov, D. A. (2016) 'New 1-D Method for Prediction of Axial-Flow Compressors Off-Design Performance', *Procedian Engineering*, 150, pp.150-160.
- [32] Koff, S. and Greitzer, E. M. (1986) 'Axisymmetrically stalled flow performance for multistage axial compressors', *Journal of Turbomachinery*, 108(2), pp.216-223.
- [33] Greitzer, E. M. (1976) 'Surge and Rotating Stall in Axial Flow Compressor, Part I', *Journal of Engineering for Power*, 98(2), pp.190-198.
- [34] Moore, F. K. and Greitzer, E. M. (1986) 'A Theory of Port-Stall Transients in Axial Compression Systems: Part I - Development of Equations', *Journal of Engineering for Gas Turbine and Power*, 108(1), pp.68-76.
- [35] Greitzer, E. M. and Moore, F. K. (1986) 'A Theory of Post-Stall Transients in Axial Compression Systems Part II - Application', *Journal of Engineering for Gas Turbine and Power*, 108(2), pp.231-239.
- [36] Al-Daini, A. J. (1986) 'Loss and Deviation Model for a Compressor Blade Element', *International Journal of Heat & Fluid Flow*, 7(1), pp.69-78.
- [37] Petrovic, M.V., Wiedermann, A. and Banjac, M.B. (2010) 'Development and validation of a new universal through flow method for axial compressors', *Proceedings of ASME Turbo Expo 2009: Power for Land, Sea, and Air*, Orlando, Florida, June 8-12, New York: ASME, GT2009-59938, 7, pp.579-588.
- [38] Pachidis, V., Templalexis, I., Pilidis, P. and Kotsiopoulos, P. (2010) 'A Dynamic Convergence Control Scheme for the Solution of the Radial Equilibrium Equation in Through-flow Analysis', *Journal of Aerospace Engineering*, 224(7), pp.803-815.

-
- [39] Hendricks, G. J., Sabnis, J. S. and Feulner, M. R. (1997) 'Analysis of Instability Inception in High Speed Multi-stage Axial-Flow Compressors', *Journal of Turbomachinery*, 199(4), pp.714-722.
- [40] Lindau, J. W. and Owen, A. K. (1997) 'Nonlinear Quasi-Three Dimensional Modelling of Rotating Stall and Surge', *33rd Joint Propulsion Conference and Exhibit*, Seattle, Washington, July 6-9, AIAA, AIAA-97-2772, pp.2772-2783.
- [41] Longley, J. P. (2007) 'Calculating Stall and Surge Transients', *Proceeding of GT2007 ASME Turbo Expo 2007: Power for Land, Sea and Air*, Montreal, Canada, May 14-17, New York: ASME, GT2007-27378, pp.125-136.
- [42] Wilson, A. G. (1996) *Stall and Surge in Axial Flow Compressors*, Phd Thesis, Cranfield University.
- [43] Gong, Y. (1999) *A Computational Model for Rotating Stall and Inlet Distortions in Multi-stage Compressors*, PhD Thesis, Massachusetts Institute of Technology.
- [44] Brand, M. L. (2013) *An Improved Blade Passage Model for Estimating Off-Design Axial Compressor Performance*, MSc Thesis, Massachusetts Institute of Technology,
- [45] Gallimore, S. J. (1998) 'Viscous Throughflow Modelling of Axial Compressor Bladerows Using a Tangential Blade Force Hypothesis', *Journal of Turbomachinery*, 120(4), pp.662-670.
- [46] Escuret, J. F., Garnier, V. (1994) 'Numerical Simulations of Surge and Rotating Stall in Multi-Stage Axial-Flow Compressors', *30th Joint Propulsion Conference and Exhibit*, Indianapolis, Indiana, June 27-29, AIAA, AIAA-94-3202, pp.3202-3211.
- [47] Longley, J. P. (1997) 'Calculating the Flowfield Behaviour of High-Speed Multi-Stage Compressors', *Proceeding of international Gas Turbine & Aero-engine Congress & Exhibition*, Orlando, Florida, June 2-5, New York:ASME, 97-GT-468, 1, pp.1-10.

- [48] Dodds, J. and Vahdati, M. (2015) ‘Rotating Stall Observations in a High Speed Compressor—Part II: Numerical Study’, *Journal of Turbomachinery*, 137(5), pp.1-10.
- [49] Haake, M., Fiola, R. and Staudacher, S. (2011) ‘Multistage Compressor and Turbine Modelling for the Prediction of the Maximum Turbine Speed Resulting from Shaft Breakage’, *Journal of Turbomachinery*, 133(2), pp.1-12.
- [50] Plemmons, L.W., Proctor, R., Albens, R. J. and Gardner, D.L. (1993) *US5205115A: Gas turbine engine case counterflow thermal control*, United States Patent and Trademark Office. Available at <https://www.google.com/patents/US5205115>.
- [51] Roe, P. L. (1981) ‘Approximate Riemann Solver, Parameter Vectors, and Difference Schemes’, *Journal of Computation Physics*, 43(2), pp.357-372.
- [52] Toro, E. F. (2007) *Riemann Solvers and Numerical Methods for Fluid dynamics*, 3rd Edition, London: Springer.
- [53] Di Mare, L. (2008) ‘Modelling of Variables Gas Properties and Compositions in Turbomachinery Flow Simulations’, *Proceedings of ASME Turbo Expo 2008: Power for Land, Sea and Air*, Berlin, Germany, June 9-13, New York: ASME, GT2008-51085, pp.2523-2533.
- [54] Mohanraj, R., Neumerier, Y. and Zinn, B. (1996) ‘Modification of Roe’s Riemann Solver for the Euler Equations with Source Terms’, *AIAA Aerospace Science Meeting and Exhibit*, Reno, Nevada, January 15-18, AIAA, AIAA-97-0766, pp.766-778.
- [55] Harten, A. and Hyman, J. H. (1983) ‘Self Adjusting Grid Methods for One-dimensional Hyperbolic Conservation Laws’, *Journal of Computational Physics*, 50(2), pp.235-269.
- [56] Eastland, A. H. J. (1982) *An Experimental Investigation of Compressor Performance in Rotating Stall* MSc Thesis, Massachusetts institute of Technology.

-
- [57] Gamache, R. N. (1985) *Axial Compressor Reversed Flow Performance*, PhD thesis, Massachusetts Institute of Technology.
- [58] Day, I. J., (1994) ‘Axial compressor performance during surge’, *Journal of Propulsion and Power*, 10(3), pp.329-336.
- [59] Hirsch, C. (1990) *Numerical Computation of Internal and External Flows, Volume 2: Computational Methods for Inviscid and Viscous Flows*, Chichester, England: Jhon Wiley and sons.
- [60] Aungier, R. H. (2003) *Axial flow compressors: a strategy for aerodynamic design and analysis*, New York: ASME Press.
- [61] Urasek, D. C., Gorrell, W. T. and Cunnan, W. S. (1979) *Technical Paper 1493: Performance of Two-Stage Fan Having Low-Aspect-Ratio, First-Stage Rotor Blading*, Cleveland, Ohio: NASA.
- [62] Moses, H. L. and Thomason, S. B. (1986) ‘Approximation for Fully Stalled Cascades’, *Journal of Propulsion and Power*, 2(2), pp.188-189.
- [63] Poinso, T. J. and Kele, D. K. (1992) ‘Boundary Conditions for Direct Simulations of Compressible Viscous Flow’, *Journal of Computational Physics*, 101(1), pp.104-129.
- [64] Haynes, J. M., Hendricks, G. J. and Epstein, A. H. (1993) ‘Active Stabilization of Rotating Stall in a Three-stage Axial Compressor’, *International Gas Turbine and Aeroengine Congress and Exposition*, Cincinnati, Ohio, May 24-27, New York: ASME, 93-GT-346, pp.1-17.
- [65] Moore, R. D. and Reid, L. (1980) *NASA Technical Paper 1659: Performance of Single-Stage Axial Flow Transonic Compressor With Rotor and Stator Aspect Ratios of 1.19 and 1.26, Respectively, and With Design Pressure Ratio of 2.05*, Cleveland, Ohio: NASA.
- [66] MacManus, D. G., Chiereghin, N., Gil Prieto, D. and Zachos, P. (2017) ‘Complex aero-engine intake ducts and dynamic distortion’, *AIAA Journal*, 55(7), pp.2395-2409.

-
- [67] Celik, I. and Karatekin, O. (1997) ‘Numerical Experiments on Application of Richardson Extrapolation With Nonuniform Grids’, *ASME Journal of Fluid Engineering*, 119(3), pp.584-590.
- [68] Celik, I. B. (2008) ‘Procedure for Estimation and Reporting of Discretization Error in CFD Applications’, *Journal of Fluids Engineering*, 130(7), pp.1-4.
- [69] Schobeiri, M. T. (1996) ‘Advanced Compressor Loss Correlations, Part I & II’, *Internal Journal of Rotating Machinery*, 3(3), pp.163-177.
- [70] ANSYS FLUENT, Release 12.0, Help System, Fluent User’s Guide, General Non-Reflecting Boundary Conditions, ANSYS Inc.
- [71] Dixon, S. L. and Hall, C. A. (2010) *Fluid mechanics and thermodynamics of turbomachinery*, 6th edition, Kidlington, England: Elsevier.
- [72] Anderson, J. D. (1995) *Computational fluid dynamics: the basics with applications*, Singapore: McGraw-Hill Education.
- [73] Tanguy, G., MacManus, D. G., Zachos, P., Gil-Prieto, D. and Garnier, E. (2017) ‘Passive Flow Control Study in an S-Duct Using Stereo Particle Image Velocimetry’, *AIAA Journal*, 55(6), pp. 1862-1877.

Appendices

A Formulation of the solver

The Euler equations are implemented in the solver as reported here. For clarity purposes the E,F,G fluxes are tied together with red, blue and green parenthesis.

Density

$$\begin{aligned} \frac{\partial \rho}{\partial t} \cdot V_{i,j} = & \left((\rho u_{\perp})_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H - (\rho u_{\perp})_{i,j,k}^{RS} \cdot A_{i,j}^H \right) + \left(((\rho u_{\theta})_{i,j,k-1}^{RS} - (\rho u_{\theta})_{i,j,k}^{RS}) \cdot A_{i,j}^{\theta} \right) \\ & + \left((\rho u_x)_{i-1,j,k}^{RS} \cdot A_{i-1,j}^V - (\rho u_x)_{i,j,k}^{RS} \cdot A_{i,j}^V \right) \end{aligned} \quad (\text{A.1})$$

Radial momentum

$$\begin{aligned} \frac{\partial \rho u_r}{\partial t} \cdot V_{i,j} = & \left((\rho u_{\perp} u_r)_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H - (\rho u_{\perp} u_r)_{i,j,k}^{RS} \cdot A_{i,j}^H \right) \\ & + \left((\rho u_{\theta} u_r)_{i,j,k-1}^{RS} - (\rho u_{\theta} u_r)_{i,j,k}^{RS} \right) \cdot A_{i,j}^{\theta} \\ & + \left((\rho u_x u_r)_{i-1,j,k}^{RS} \cdot A_{i-1,j}^V - (\rho u_x u_r)_{i,j,k}^{RS} \cdot A_{i,j}^V \right) \\ & + \left(P_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H \cdot \cos(\lambda_{A,i,j-1}) - P_{i,j,k}^{RS} \cdot A_{i,j}^H \cdot \cos(\lambda_{A,i,j}) \right) + \frac{(\rho u_{\theta}^2)_{i,j,k}}{r_{i,j}} V_{i,j} \end{aligned} \quad (\text{A.2})$$

Circumferential momentum

$$\begin{aligned} \frac{\partial \rho u_{\theta}}{\partial t} \cdot V_{i,j} = & \left((\rho u_{\perp} u_{\theta})_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H - (\rho u_{\perp} u_{\theta})_{i,j,k}^{RS} \cdot A_{i,j}^H \right) \\ & + \left((\rho u_{\theta}^2 + P)_{i,j,k-1}^{RS} - (\rho u_{\theta}^2 + P)_{i,j,k}^{RS} \right) \cdot A_{i,j}^{\theta} \\ & + \left((\rho u_x u_{\theta})_{i-1,j,k}^{RS} \cdot A_{i-1,j}^V - (\rho u_x u_{\theta})_{i,j,k}^{RS} \cdot A_{i,j}^V \right) - \frac{(\rho u_{\theta} u_r)_{i,j,k}}{r_{i,j}} V_{i,j} \end{aligned} \quad (\text{A.3})$$

Axial momentum

$$\begin{aligned} \frac{\partial \rho u_x}{\partial t} \cdot V_{i,j} = & \left((\rho u_{\perp} u_x)_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H - (\rho u_{\perp} u_x)_{i,j,k}^{RS} \cdot A_{i,j}^H \right) \\ & + \left((\rho u_{\theta} u_x)_{i,j,k-1}^{RS} - (\rho u_{\theta} u_x)_{i,j,k}^{RS} \right) \cdot A_{i,j}^{\theta} \\ & + \left((\rho u_x^2 + P)_{i-1,j,k}^{RS} \cdot A_{i-1,j}^V - (\rho u_x^2 + P)_{i,j,k}^{RS} \cdot A_{i,j}^V \right) \\ & - \left(P_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H \cdot \sin(\lambda_{A,i,j-1}) - P_{i,j,k}^{RS} \cdot A_{i,j}^H \cdot \sin(\lambda_{A,i,j}) \right) \end{aligned} \quad (\text{A.4})$$

Energy equation

$$\begin{aligned} \frac{\partial \rho E_0}{\partial t} \cdot V_{i,j} = & \left((\rho u_{\perp} H)_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H - (\rho u_{\perp} H)_{i,j,k}^{RS} \cdot A_{i,j}^H \right) \\ & + \left((\rho u_{\theta} H)_{i,j,k-1}^{RS} - (\rho u_{\theta} H)_{i,j,k}^{RS} \right) \cdot A_{i,j}^{\theta} \\ & + \left((\rho u_x H)_{i-1,j,k}^{RS} \cdot A_{i-1,j}^V - (\rho u_x H)_{i,j,k}^{RS} \cdot A_{i,j}^V \right) \end{aligned} \quad (\text{A.5})$$

Where the variables with the superscript RS are calculated based on the result of the Riemann Solver at the face that they are multiplied by. The parts with no superscript (time derivative and source terms) are calculated using values in the volumes.

In the rotors and stator the equations are modified by adding body forces, removing circumferential fluxes and introducing translation of the variables with the blades (in magenta brackets).

Density

$$\begin{aligned} \frac{\partial \rho}{\partial t} \cdot V_{i,j} = & \left((\rho u_{\perp})_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H - (\rho u_{\perp})_{i,j,k}^{RS} \cdot A_{i,j}^H \right) \\ & - \frac{\Omega}{6\Delta\theta} \left(2\rho_{i,j,k+1} + 3\rho_{i,j,k} - 6\rho_{i,j,k-1} + \rho_{i,j,k-2} \right) \\ & + \left((\rho u_x)_{i-1,j,k}^{RS} \cdot A_{i-1,j}^V - (\rho u_x)_{i,j,k}^{RS} \cdot A_{i,j}^V \right) \end{aligned} \quad (\text{A.6})$$

Radial momentum

$$\begin{aligned} \frac{\partial \rho u_r}{\partial t} \cdot V_{i,j} = & \left((\rho u_{\perp} u_r)_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H - (\rho u_{\perp} u_r)_{i,j,k}^{RS} \cdot A_{i,j}^H \right) \\ & - \frac{\Omega}{6\Delta\theta} \left(2(\rho u_r)_{i,j,k+1} + 3(\rho u_r)_{i,j,k} - 6(\rho u_r)_{i,j,k-1} + (\rho u_r)_{i,j,k-2} \right) \\ & + \left((\rho u_x u_r)_{i-1,j,k}^{RS} \cdot A_{i-1,j}^V - (\rho u_x u_r)_{i,j,k}^{RS} \cdot A_{i,j}^V \right) \\ & + \left(P_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H \cdot \cos(\lambda_A)_{i,j-1} - P_{i,j,k}^{RS} \cdot A_{i,j}^H \cdot \cos(\lambda_A)_{i,j} \right) \\ & + \frac{(\rho u_{\theta}^2)_{i,j,k}}{r_{i,j}} V_{i,j} + (F_r)_{i,j,k} \cdot V_{i,j} \end{aligned} \quad (\text{A.7})$$

Circumferential momentum

$$\begin{aligned}
\frac{\partial \rho u_\theta}{\partial t} \cdot V_{i,j} = & \left((\rho u_\perp u_\theta)_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H - (\rho u_\perp u_\theta)_{i,j,k}^{RS} \cdot A_{i,j}^H \right) \\
& - \frac{\Omega}{6\Delta\theta} \left(2(\rho u_\theta)_{i,j,k+1} + 3(\rho u_\theta)_{i,j,k} - 6(\rho u_\theta)_{i,j,k-1} + (\rho u_\theta)_{i,j,k-2} \right) \\
& + \left((\rho u_x u_\theta)_{i-1,j,k}^{RS} \cdot A_{i-1,j}^V - (\rho u_x u_\theta)_{i,j,k}^{RS} \cdot A_{i,j}^V \right) - \frac{(\rho u_\theta u_r)_{i,j,k}}{r_{i,j}} V_{i,j} \\
& + (F_\theta)_{i,j,k} \cdot V_{i,j}
\end{aligned} \tag{A.8}$$

Axial momentum

$$\begin{aligned}
\frac{\partial \rho u_x}{\partial t} \cdot V_{i,j} = & \left((\rho u_\perp u_x)_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H - (\rho u_\perp u_x)_{i,j,k}^{RS} \cdot A_{i,j}^H \right) \\
& - \frac{\Omega}{6\Delta\theta} \left(2(\rho u_x)_{i,j,k+1} + 3(\rho u_x)_{i,j,k} - 6(\rho u_x)_{i,j,k-1} + (\rho u_x)_{i,j,k-2} \right) \\
& + \left((\rho u_x^2 + P)_{i-1,j,k}^{RS} \cdot A_{i-1,j}^V - (\rho u_x^2 + P)_{i,j,k}^{RS} \cdot A_{i,j}^V \right) \\
& - \left(P_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H \cdot \sin(\lambda_A)_{i,j-1} - P_{i,j,k}^{RS} \cdot A_{i,j}^H \cdot \sin(\lambda_A)_{i,j} \right) + (F_x)_{i,j,k} \cdot V_{i,j}
\end{aligned} \tag{A.9}$$

Energy equation

$$\begin{aligned}
\frac{\partial \rho E_0}{\partial t} \cdot V_{i,j} = & \left((\rho u_\perp H)_{i,j-1,k}^{RS} \cdot A_{i,j-1}^H - (\rho u_\perp H)_{i,j,k}^{RS} \cdot A_{i,j}^H \right) \\
& - \frac{\Omega}{6\Delta\theta} \left(2(\rho E_0)_{i,j,k+1} + 3(\rho E_0)_{i,j,k} - 6(\rho E_0)_{i,j,k-1} + (\rho E_0)_{i,j,k-2} \right) \\
& + \left((\rho u_x H)_{i-1,j,k}^{RS} \cdot A_{i-1,j}^V - (\rho u_x H)_{i,j,k}^{RS} \cdot A_{i,j}^V \right) + (W_{EX})_{i,j,k} \cdot V_{i,j}
\end{aligned} \tag{A.10}$$

Where Ω and W_{EX} are 0 for stators. The indices k for the rotational term $k+1, k, k-1, k-2$ are valid only for positive Ω , for negative rotations they are $k-1, k, k+1, k+2$.

B Formulation of the reverse flow deviation correlation

A new correlation has been developed to estimate the deviation of a blade passage operating in reverse flow. This is based on the model proposed by Longley described in section 4.7 of which it is a lumped version. The flow non-uniformity b can be estimated from the inlet flow based on equation 4.52 and the deviation can be calculated from it with equation 4.53. In this appendix is described how the correlation is modified to introduce the presence of mixing inside of the blade.

The axil position at which the maximum separation (i.e. the b calculated with equation 4.52) is reached is assumed to be at $1/4$ of the chord from the T.E. and from this point the profile along the passage of b is simulated based on the prescribed ratio. We can then estimate the value of b at the L.E., from which the angle after the mixing can be calculated.

If equations 3.24 and 3.25 are put together and the transport terms are cancelled the result is

$$\frac{\partial}{\partial t} \left(\frac{1}{2} \rho (b-1) \bar{u}^2 \right) = - \left(\frac{b|\bar{u}|}{0.5s} \right) \frac{1}{2} \rho (b-1) \bar{u}^2 . \quad (\text{B.1})$$

Therefore the rate at which the excess energy is mixed out is instantaneously equal to the excess energy over the time in which the jet travels through the distance of half a pitch. The decay of excess energy is modelled in the present work by modifying equation B.1 into a space derivative:

$$\frac{\partial}{\partial x} \left(\frac{1}{2} \rho (b-1) \bar{u}^2 \right) = - \frac{1}{0.5s} \frac{1}{2} \rho (b-1) \bar{u}^2 . \quad (\text{B.2})$$

If the chord true length is divided into n elements of length $dx_c = c/n$ the excess energy at any element i can be calculated from the previous $i-1$ as

$$\left(\frac{1}{2} \rho (b-1) \bar{u}^2 \right)_i = \left(\frac{1}{2} \rho (b-1) \bar{u}^2 \right)_{i-1} - \frac{\partial}{\partial x} \left(\frac{1}{2} \rho (b-1) \bar{u}^2 \right)_{i-1} \cdot dx_c . \quad (\text{B.3})$$

Substituting the derivative the value of the excess energy at the last point can

be then written as

$$\left(\frac{1}{2}\rho(b-1)\bar{u}^2\right)_n = \left(\frac{1}{2}\rho(b-1)\bar{u}^2\right)_1 \cdot \left(1 - \frac{dx_c}{0.5s}\right)^n. \quad (\text{B.4})$$

The accuracy of this approximation increases with the number of elements in which the chord is divided, however the computational time is the same whatever n is used. A value of n of 1000 is used as the function is *converged* for that value, i.e. the result is practically the same for any n higher than that. As it was assumed that the separation is reached at 1/4 of the chord the c used is 3/4 of the full true chord. By making the assumption of incompressibility the density is considered constant along the chord, and therefore the value of b at the flow outlet can be obtained directly from the excess energy.

The rate of wake mixing out is chosen from Longley as an arbitrary value with the justification that whatever is the length of the wake the integral of the entropy creation is the same. While this is true for the pressure loss the rate of mixing does have an effect on the deviation. In fact from equation 4.53 it is clear that the final value of the flow angle depends on the value of b at the trailing edge, and this in turn depends on the mixing ratio. The value of mixing ratio shown in the equation based on half a pitch was found to predict excessively low deviation. A value of twice a pitch, $2s$ was found to give good results for 3 different compressors as will be reported in chapter 6. Note that the value proposed by Longley was $s/2$, four times smaller than the current one.

The deviation angle is then $\delta = f(\alpha_{IN}, \gamma, \sigma)$ As the deviation is calculated from the inlet angles it can not be estimated if the flow is exiting from both blade edges, in this case the flow exit angle is set as the metal outlet angle ($\delta = 0$).

C Formulation of the non-reflective boundary condition

The terms d_1, d_2, d_3, d_4, d_5 contain the incoming and outgoing characteristic waves amplitude L . They are calculated as:

$$\begin{aligned} d_1 &= \frac{1}{a^2} \left(L_2 + \frac{1}{2} (L_5 + L_1) \right) \\ d_2 &= \frac{1}{2} (L_5 + L_1) \\ d_3 &= \frac{1}{2\rho a} (L_5 - L_1) \\ d_4 &= L_3 \\ d_5 &= L_4 \end{aligned} \tag{C.1}$$

where the wave amplitudes L are:

$$\begin{aligned} L_1 &= \lambda_1 \left(\frac{\partial P}{\partial x} - \rho a \frac{\partial u_x}{\partial x} \right) \\ L_2 &= \lambda_2 \left(a^2 \frac{\partial \rho}{\partial x} - \frac{\partial P}{\partial x} \right) \\ L_3 &= \lambda_2 \frac{\partial u_\theta}{\partial x} \\ L_4 &= \lambda_2 \frac{\partial u_r}{\partial x} \\ L_5 &= \lambda_5 \left(\frac{\partial P}{\partial x} + \rho a \frac{\partial u_x}{\partial x} \right) \end{aligned} \tag{C.2}$$

The λ are the velocities of the characteristic waves, which are the eigenvalues of the Euler equations:

$$\lambda_1 = u_x - a \tag{C.3}$$

$$\lambda_2 = u_x$$

$$\lambda_5 = u_x + a$$

$$\tag{C.4}$$

The derivative are calculated using the values in the 2 axial position closest at the boundary, for example for the element of index N_x, j, k the derivative is calculated as:

$$\frac{\partial P}{\partial x_{N_x, j, k}} = \frac{P_{N_x, j, k} - P_{N_x-1, j, k}}{dx} \quad (C.5)$$

Similarly for the position $i = 1$ the elements used are $i = 1, 2$

The equations C.2 are the basic characteristics amplitude, when used in the boundary conditions they have the same behaviour of extrapolated boundary conditions. This set of equations is used for supersonic flow, when nothing is imposed.

For subsonic flow exiting the domain the static pressure P_{BC} is imposed, this is done by modifying the first amplitude as

$$L_1 = K_W(P_{n, j, k} - P_{BC}) \quad (C.6)$$

For subsonic flow entering the domain the direction, the total temperature and total pressure must be imposed. From the prescribed total properties and the inlet velocity the static value of pressure and temperature can be obtained, T_{BC} , P_{BC} . The tangential and radial components of velocity instead are prescribed to be zero when the flow is entering. The wave amplitudes for this case are calculated as:

$$L_1 = K_W(P - P_{BC}) \quad (C.7)$$

$$L_2 = K_W \left(a^2 \left(\rho - \frac{P_{BC}}{R T_{BC}} \right) - (P - P_{BC}) \right)$$

$$L_3 = K_W \cdot u_\theta$$

$$L_4 = K_W \cdot u_r$$

$$L_5 = \lambda_5 \left(\frac{\partial P}{\partial x} + \rho a \frac{\partial u_x}{\partial x} \right)$$

The parameter K_W determines how quickly the boundary will return to the prescribed values after a wave has exited the domain (see figure 4.23). It is defined as:

$$K_W = \sigma \cdot a \quad (C.8)$$

were the constant σ is an under relaxation factor, the value of 5 is proposed as default in ANSYS FLUENTTM user manual [70]. This has been found to be small enough to properly dampen any reflection and return the boundary conditions to the prescribed value in a reasonable time after a wave passage. For large values of time step dt a lower value of σ is necessary as the readjustment can be so fast that it overshoots and makes solver unstable. This has been observed in some test cases with large element size and therefore large time steps.

D Code flow chart

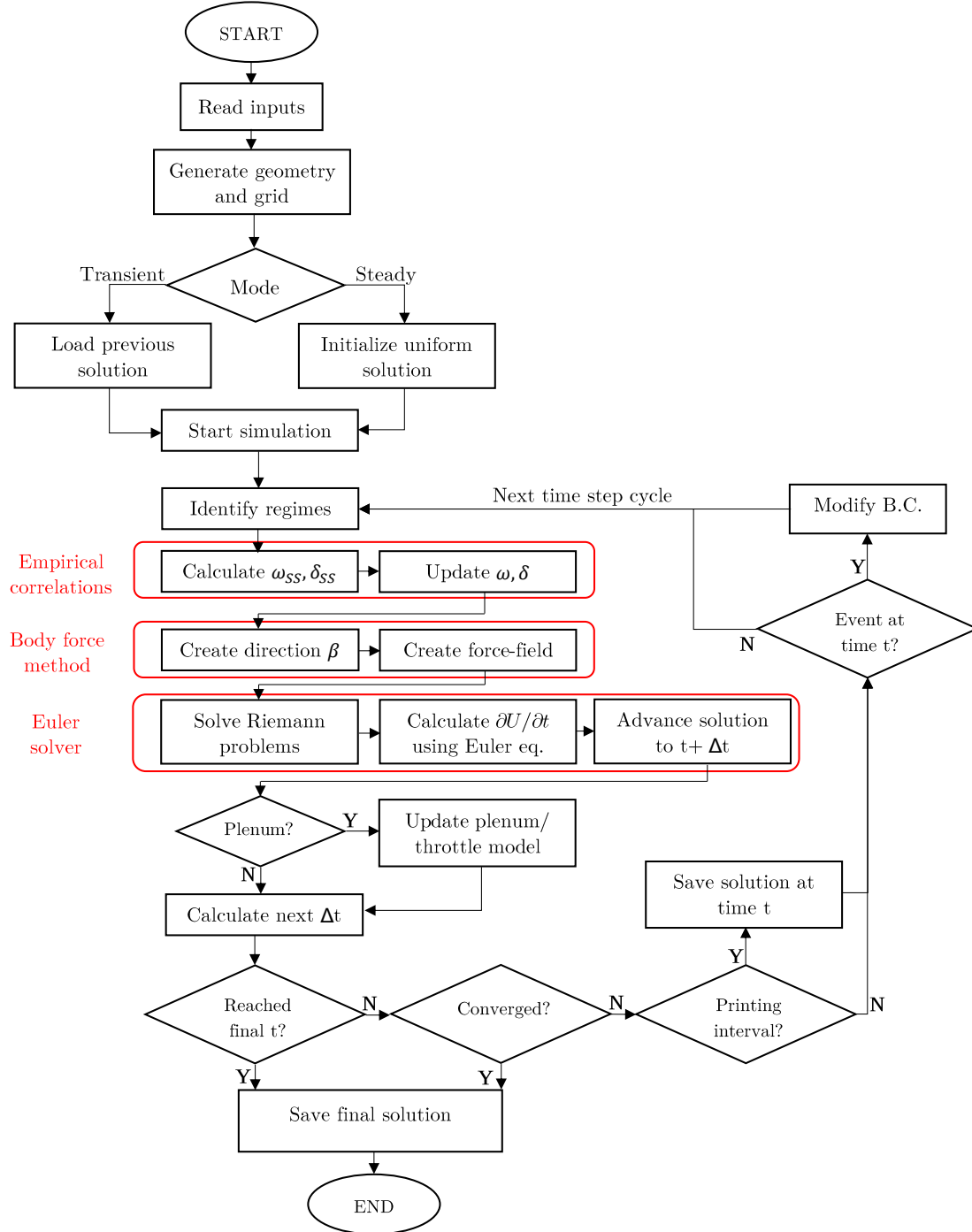


Fig. D.1: Code flow chart.