

GT2025-153790

A NEW BODY FORCE-BASED MODELLING APPROACH FOR CENTRIFUGAL COMPRESSOR POST-STALL AERODYNAMICS

Dimitrios Lamprakis^{1,*}, Mauro Righi¹, Vassilios Pachidis¹

¹Rolls-Royce University Technology Centre, Cranfield University, MK43 0AL, United Kingdom

ABSTRACT

A lower-order modelling approach is developed, whereby the compression system is modelled as an empty duct, with body force-fields imparting turning and losses to the flow. The flow-field solution is obtained transiently by solving the axisymmetric Euler equations with metal and aerodynamic blockage in the relative frame of reference. The solver is coupled with a fully custom body force model consisting of a base turning force, correction force and viscous force component. A new blockage-mixing model is derived providing estimations for aerodynamic blockage as well as the associated mixing losses and change in flow direction. Separation losses are introduced via the turning force coupled with a shape function which allows adjustment of the reattachment position. The blockage-mixing and separation modelling methods are verified against 3D RANS data, providing consistently reliable predictions. The steady-state, reverse flow performance predictions at different speeds are compared against 3D RANS simulations, showing excellent agreement. Surge simulations are subsequently carried out using an 1D plenum model. As the experimental rig's plenum details are not provided, the plenum volume is varied until the experimentally reported surge cycle frequency is achieved. Unsteady pressure measurements are compared against the experimental ones, showing excellent agreement in the entire range of the surge cycles. Flow-field contours at different time instances are in concert with the experimentally obtained DPIV measurements, accurately capturing the onset of stall in the vaned diffuser. The simulations indicate deep surge behaviour, confirming conjectures from the experimental report which could not eventually be confirmed during testing. The code constitutes the first successful, validated, through-flow approach, capable of predicting the post-stall, steady-state and transient aerodynamic performance in centrifugal compressors, while reducing the computational cost by at least one order of magnitude compared to conventional CFD modelling approaches.

Keywords: Body force, through-flow, blockage, mixing, cen-

trifugal compressor, RANS, stall, surge

NOMENCLATURE

b	Blockage parameter
C	Distribution coefficient
E_r	Specific relative internal energy
$\mathbf{F} = (F_x, F_\theta, F_r)$	Body force vector
F_{cor}	Correction force
F_{turn}	Turning force
F_{visc}	Viscous force
I	Rothalpy
K_{cor}	Correction force scaling coefficient
L_c	Camberline length
m	Meridional coordinate
$\dot{m}_{cor}$	Corrected mass flow
$\dot{m}$	Mass flow
n	Spanwise coordinate
P	Static pressure
$P_{0,\Omega}$	Rotary stagnation pressure
r	Radial coordinate
R	Ideal as constant
t	Time
T	Static temperature
$\mathbf{W} = (w_x, w_\theta, w_r)$	Velocity vector in the relative FoR, expressed in cylindrical coordinates
V_{mer}	Meridional velocity magnitude
$\mathbf{V} = (v_x, v_\theta, v_r)$	Velocity vector in the absolute FoR, expressed in cylindrical coordinates
α	Absolute flow angle with respect to meridional direction
β	Relative flow angle with respect to meridional direction
γ	Specific heat ratio
δ	Deviation angle
θ	Circumferential coordinate
κ	Blade metal angle with respect to meridional direction
λ	Lean angle
Λ	Arbitrary scalar property
ρ	Density
τ	Torque

*Corresponding author: Dimitrios.Lamprakis@cranfield.ac.uk

φ	Flow path angle
Ω	Angular velocity

Abbreviations

<i>CFD</i>	Computational fluid dynamics
<i>FoR</i>	Frame of reference
<i>LE</i>	Leading edge
<i>PR_{tt}</i>	Total-to-total pressure ratio
<i>RANS</i>	Reynolds-averaged Navier-Stokes
<i>SB</i>	Splitter blade
<i>ss</i>	Suction surface
<i>TE</i>	Trailing edge

Subscripts

<i>abs</i>	Property in the absolute FoR
<i>act</i>	Actual/measured property
<i>CFD</i>	Property extracted from CFD simulations
<i>cor</i>	Property related to correction force
<i>imp</i>	Imposed property
<i>pl</i>	Property related to the plenum
<i>rel</i>	Property in the relative FoR
<i>stream</i>	Streamwise property
<i>visc</i>	Property related to viscous force

Superscripts

*	Related to numerical flow slippage / modified quantity
–	Averaged property

1. INTRODUCTION

The operation of aero-engines is limited in a way to prevent the manifestation, propagation and growth of compressor flow instabilities. These are known as rotating stall and surge and entail significant implications on the structural integrity and operability of the engine. These instabilities are typically triggered by inlet flow distortions [1], blade deterioration [2] and rapid transients [3], while in seldom occasions due to shaft failure and errors in the engine control logic.

Compressor instabilities and their inception mechanisms are up to this date not fully understood, while the knowledge gap is exacerbated when considering centrifugal configurations [4, 5]. Experimental campaigns have traditionally been the most reliable method for the investigation of compressor stall behaviour, however this is only possible at advanced/mature design phases and entail excessive costs, while the tests are typically restricted to low rotational speeds to prevent severe structural damage. Conventional CFD modelling methods provide an alternative approach but the associated computational cost requirements are often excessive. Fidalgo et al. [6] for instance, investigated the response of NASA's stage 67 at non-uniform inlet conditions using uRANS simulations. Convergence was reached within 20-30 rotor revolutions, corresponding to a computational time of 1-2 months on a 128 core cluster. Zhao et al. [7] investigated the post-stall transient response of a modern, multi-stage axial compressor, via full annulus, uRANS simulations. The computational domain consisted of approximately 300×10^6 elements and a shaft revolution required one day on 192 cores. Shahin et al. [8] investigated the primary flow structures of the NASA CC3 compressor during a surge cycle using large eddy simulations (LES). Simulation of a single surge cycle on a 120° sector consisting of 9.7m elements, required 3 months on a 48-core processor.

Through-flow methods provide a computationally efficient alternative, replacing the turbomachinery components with body forces. Gong was the first to demonstrate the ability of through-flow methods to simulate inlet flow distortions, rotating stall and spike-type stall inception in a multi-stage axial compressors, but no reverse flow capabilities are reported [9]. Incorporating Gong's body force model into a 3D RANS code, Hsiao et al. [10] modelled NASA's 22" rig. The coefficients for the model were derived via 3D, single passage, RANS simulations. The code accurately reproduced the experimental total pressure, temperature and swirl angle spanwise profiles as well as the separation angle of attack at the intake. Chima et al. [11] simulated the NASA stage 35 (axial compressor stage) using a 3D RANS solver and combining Stewart's [12] and Marble's [13] body force models depending on the operating conditions. Inlet flow distortions were analysed, alongside the simulation of onset, evolution, and recovery phases of rotating stall events. Using a custom, axisymmetric, inviscid solver with blockage and body forces, Escuret and Garnier [14] successfully predicted the onset of surge in an axial compressor although the surge event itself was not simulated. Rotating stall was also simulated by solving multiple meridional flow fields within bladed regions, but was not validated against experimental data. Stewart [12] developed an iterative-based body force approach, simulating the flow path of an entire turbofan engine using axisymmetric RANS simulations. Longley [15] developed a 3D code aiming to simulate transient post-stall aerodynamics of multi-stage axial compressors with body forces. He introduced a blockage parameter to model small-lengthscale phenomena along with an additional equation which governs its dissipation and generation. The code was validated against low-speed, multi-stage axial compressors. More recently, Righi et al. [16–18] created a 3D, through-flow code (ACRoSS) with body forces to simulate surge and rotating stall in axial compressors, validating the tool against multi-stage, low-speed compressors and advanced, high-speed designs.

The applicability of the aforementioned studies was strictly demonstrated to axial compressors. Through-flow methods for centrifugal compressors are very scarce in the literature. Kotapalli [19] developed a streamline, curvature-based body force model for centrifugal compressors and incorporated it into a commercial CFD code. Key properties such as the deviation and meridional entropy gradient were extracted from single passage RANS simulations. A 2D, radial impeller, was successfully simulated at steady-state, forward-flow conditions but instabilities were encountered in a 3D geometry. Benneke [20] extracted the body forces directly from 3D, single-passage RANS simulations at different rotational speeds and operating conditions, formulating a body force database. Steady-state operating conditions were accurately obtained only at 75% speed due to convergence issues and limitations of the look-up table approach at higher rotational speeds and transonic flow regimes. Stall inception due to both modal waves and spikes was demonstrated. Backwards-travelling modal disturbances were captured at 75% design speed, the presence of which, was recorded experimentally at 100% design speed by Spakovszky [21]. No post-stall response i.e. rotating stall or surge capabilities are reported from either authors. Powers et al. [22, 23] developed a reduced-order, 1D model using ordinary

differential equations to describe impeller and diffuser dynamics. A blockage function estimated the aerodynamic blockage, with skin friction losses as the sole reverse flow loss mechanism. The model, reliant on experimentally determined coefficients, showed good accuracy at low speeds but larger errors at higher speedlines. Using a 1D throttle model, it realistically simulated unsteady behaviour near and beyond the surge line and accurately predicted steady-state reverse flow characteristics at various rotational speeds [22]. No body-force-based method for centrifugal compressor post-stall is reported in the literature.

The first objective of the present paper is to propose a generalised, body force-based approach for turbomachinery, post-stall aerodynamic applications, focusing on centrifugal compressors. Expanding on novel through-flow framework of Lamprakis et al. [24], a complete, physics-based, lower-order model is proposed, capable of dealing with unstalled and highly-stalled flow and estimating key aerodynamic cascade quantities, i.e. pressure losses, aerodynamic blockage and change in flow direction. The second objective of the paper is the verification of the individual lower-order models and the validation of the entire through-flow approach against publicly available experimental surge data of the NASA CC3 centrifugal compressor [25, 26]. The former is carried out by comparing predictions of the models against 3D RANS data. The latter entails an initial verification by comparing reverse flow, steady-state predictions obtained with through-flow and RANS simulations and validation of transient surge predictions against experimental data.

2. METHODOLOGY

A body force-based modelling approach for centrifugal compressor transient, post-stall aerodynamics is developed, aiming to reproduce key, long-lengthscale flow features and predict global performance compressor quantities, while drastically reducing the computational cost compared to conventional CFD methods. The developed methods are implemented in the form of a through-flow code (AXCSTAR), incorporating a meshing module, custom solver, custom body force model and a module for blade curvature and blockage definition [24]. Lower-order, physics-based methods are developed, introducing key aerothermal aspects of the different, complex flow regimes encountered during compressor stall and surge events.

2.1 Governing equations and solver

The flow is modelled as compressible, inviscid and axisymmetric. The governing equations are solved transiently in the relative frame of reference (FoR) on a 2D, structured, body force-relevant grid, representative of the compressor's annulus on the meridional plane. Godunov's finite volume scheme is implemented, coupled with HLLC solver to obtain the fluxes. The inherent axisymmetric nature of surge events enables a justified dimensionality reduction, while preserving low computational requirements. A more in-depth analysis of the solver can be found in [24]. Body force source terms (F_x, F_θ, F_r) are added to introduce flow turning and losses. These represent the component-wise summation of turning, correction, and viscous force contributions along the three principal directions. A blockage parameter, $b = b_{metal} + b_{aero}$, is introduced to model metal and aerodynamic

blockage, with the latter being varied transiently. The governing equations are given in matrix form below:

$$\begin{aligned} & \frac{\partial}{\partial t} \begin{bmatrix} b\rho \\ b\rho w_x \\ b\rho w_\theta \\ b\rho w_r \\ b\rho E_{rel} \end{bmatrix} + \frac{\partial}{\partial x} \begin{bmatrix} b\rho w_x \\ b\rho w_x^2 + P \\ b\rho w_\theta w_x \\ b\rho w_r w_x \\ b\rho w_x I \end{bmatrix} + \frac{1}{r} \frac{\partial}{\partial r} \begin{bmatrix} b\rho r w_r \\ b\rho r w_x w_r \\ b\rho r w_\theta w_r \\ b\rho r w_r^2 + Pr \\ b\rho r w_r I \end{bmatrix} \\ & + \begin{bmatrix} 0 \\ 0 \\ \frac{b\rho w_\theta w_r}{r} \\ -\frac{b(\rho w_\theta^2 + P)}{r} \\ 0 \end{bmatrix} + \begin{bmatrix} 0 \\ 0 \\ 2b\rho\Omega w_r \\ -b\rho(2\Omega w_\theta + \Omega^2 r) \\ 0 \end{bmatrix} = \begin{bmatrix} 0 \\ bF_x \\ bF_\theta \\ bF_r \\ 0 \end{bmatrix} + \begin{bmatrix} 0 \\ P \frac{\partial b}{\partial x} \\ 0 \\ P \frac{\partial b}{\partial r} \\ 0 \end{bmatrix} \end{aligned} \quad (1)$$

2.2 Body force model

The compressor is modelled as a hollow duct, introducing flow turning and entropy gradients via a turning and viscous force model respectively. The former is a function of the blade and deviation angle and is constituted by a base and correction force component, while the latter is a function of the pressure loss; at forward flow conditions, these are predicted using Aungier's [27] empirical correlations and distributed along the passage using distribution coefficients. The body force model and distribution coefficients along with its validation, limitations and implementation aspects are presented in detail in [24] and are not covered herein. The base turning force component is exerted normal to the relative flow path and is given by Eq. (2):

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

The correction force component is computed by Eq. (3):

$$F_{cor} = K_{cor} \frac{|V_{mer}| (w_{\theta,act} + \Omega r)}{L_c} \left[1 - \frac{V_{mer} \tan\beta_{imp} + \Omega r}{w_{\theta,act} + \Omega r} \right] \quad (3)$$

The correction force is exerted whenever flow misalignment occurs, i.e. prescribed and actual flow angle do not match, ensuring the accurate imposition of the prescribed flow turning, particularly in the presence of strong flow-field gradients. The total

turning force is then the summation of Eq. (2) and Eq. (3). The viscous force model transforms the predicted relative stagnation pressure loss to a force acting parallel to the relative streamlines as per Eq. (4):

$$F_{visc} = \cos\beta \frac{P}{\rho} \left\{ \frac{1}{P_{0,rel}} \frac{\partial P_{0,rel}}{\partial m} - \left(\frac{\gamma \sin \varphi}{1 + \frac{\gamma-1}{2} Ma_{rel}^2} \right) \frac{Ma_{\Omega}^2}{r} \right\} \quad (4)$$

Key compressor cascade performance quantities, e.g. flow deviation and pressure losses are distributed along the blade passage using custom or CFD-derived distribution coefficients. Their extraction and different mathematical descriptions are presented in [24]. A simple, formal definition of the coefficient is given below:

$$C_{\Lambda}(m, n) = \frac{\Lambda(m, n)}{\bar{\Lambda}} \quad (5)$$

2.3 Reverse flow aerodynamics modelling

The aerodynamic performance of a compressor is predominantly determined by three aspects; the energy input into the flow and by extension the flow turning inside rotors, viscous dissipation which manifests in the form of stagnation pressure loss (and temperature rise) and the effective annulus area which ultimately dictates the mass flow capacity of the compressor and is a function of both metal and aerodynamic blockage. Depending on the direction of the flow at the leading edge (LE) and trailing edge (TE), the flow regime is characterised as: forward, mixed inwards, mixed outwards and reverse [16]. A schematic overview of the different regimes are shown in Fig. 1. The classification is based on the flow direction of the pitchwise-averaged velocity components along with a concurrent local flow direction evaluation to ensure modelling consistency when the local flow regime is inconsistent with the global passage one. A compound of empirical loss and deviation/slip correlations, analytical models and other methods are combined to determine the flow direction, losses and aerodynamic blockage parameter.

Forward flow modelling. When the ratio of the flow entering and exiting at both the LE and TE with respect to the total flow rate at these boundaries exceeds a prescribed threshold, then the flow is classed as "forward" (Fig. 1a). At these conditions, deviation/slip, pressure loss and aerodynamic blockage are estimated by empirical correlations [27] in a transient manner.

Reverse flow modelling. An identical approach is adopted to characterise reverse flow conditions, with the difference being the occurrence of inflow and outflow in the passage at the TE and LE, respectively. As no dedicated empirical correlations for reverse flow aerodynamic predictions are available in the literature, losses are introduced via the turning force, by projecting it perpendicular to the blade camberline, instead of the flow. This generates flow separation and a component parallel to the relative flow-path acting as an entropy-generation force, representative of the separation loss, which is the primary source of loss during reverse flow operation. This modelling method is used for axial compressor rotating stall and surge modelling in [15, 16]. This approach is further extended herein, by defining a shape function which enables the adjustment of the reattachment point, i.e. the

extent of the separation region. This enables control of the separated region extent and consequently, the losses. The shape of the separated region and local flow direction arise naturally from the misalignment between the relative flow path and the turning force direction. The imposed flow angle is consequently defined in a piecewise manner using the modified β_{imp}^* (by the shape function) and the actual flow imposed flow angle as per Eq. (6).

$$\beta_{imp} = \begin{cases} \beta_{imp}^* & \text{if } 0 \leq \xi < \xi_{tr} \\ \kappa(m, n) + C_{\delta}^*(m, n) \bar{\delta}|_{TE} & \text{if } \xi_{tr} \leq \xi \leq 1 \end{cases} \quad (6)$$

In the above equations, C_{δ}^* is the modified deviation angle distribution coefficient, ξ the normalised meridional camberline length, with 0 and 1 denoting the TE and LE respectively and ξ_{tr} the normalised location of the reattachment point. Figure 1 (top right) illustrates the imposed relative flow angle, assuming zero deviation, during reverse flow at midspan of an impeller blade. Three modified curvature regions are shown, extending 10%, 30% and 60% of the blade's stream length from the TE.

The implications of the highly separated flow at the TE are not limited to the losses due to flow separation; the separation bubble precipitates aerodynamic blockage in the vicinity of the blade passage TE, while the mixing of the bulk flow with the separated wake induces additional losses and change in flow direction. Modelling of all these mechanisms is essential to adequately reproduce bulk, reverse flow aerodynamic features in a through-flow code. These are estimated using an analytical model, discussed in the next section.

Mixed flow modelling. The flow regime inside a blade passage is characterised as mixed, whenever simultaneous and substantial inflow or outflow occurs at the LE and the TE or the criteria neither of forward nor reverse flow regime are met. The flow regime is further sub-categorised as mixed flow-inwards or mixed flow-outwards when the flow is simultaneously entering and exiting the blade passage, respectively. Due to the unpredictable, chaotic nature of mixed flow, the modelling approach proposed by Righi [16] is adopted. Flow deviation is neglected, while losses are purely introduced via the turning force model. In the case of mixed inwards flow, reattachment positions downstream the LE and the TE are specified.

2.4 Blockage-mixing modelling

During flow reversal, the high incidence at the TE causes large separation bubbles occupying large fractions of the blade passage. In axial compressor passages, these are mostly uniform in size and shape, with most of the mixing occurring once the flow exits the passage. In impeller and vaned diffuser blades, the larger streamwise length of the blades increases the residence time of the flow inside the passages, enabling a higher degree of mixing inside the passage, progressively reducing the bubble size. The mixing process also precipitates losses along with change in the direction of the flow.

The blockage-mixing model is formed on the basis of the coexistence of two distinct flow streams with the first representing the bulk flow and the latter the blocked region due to the presence of a separation bubble. It also assumes that mixing

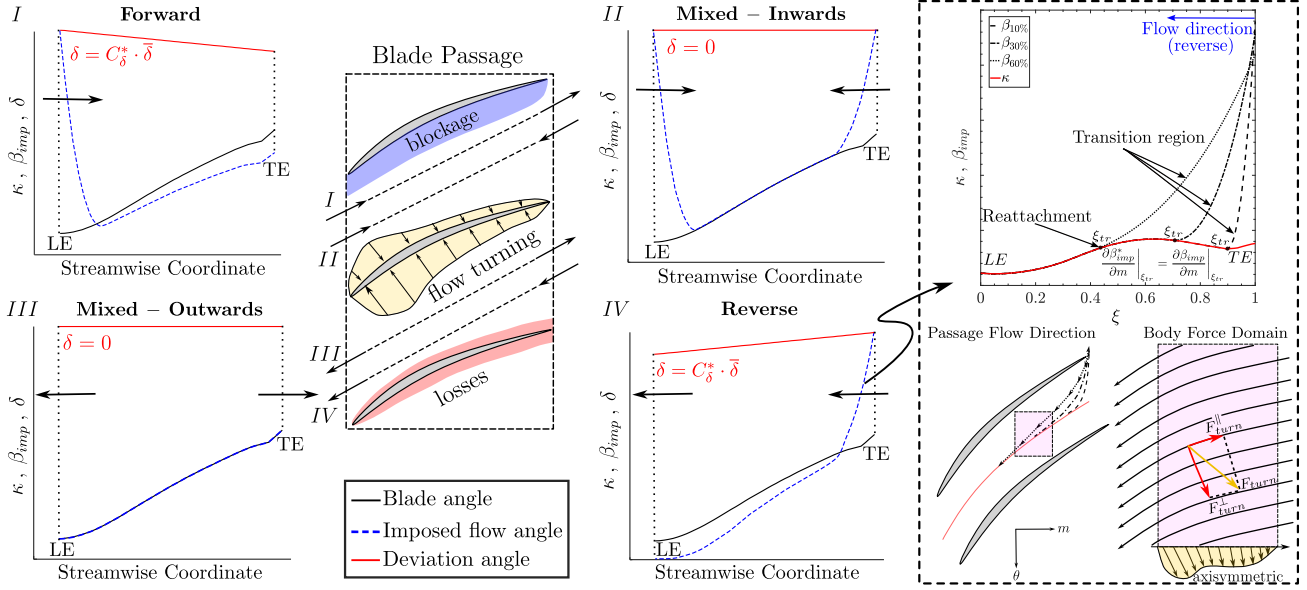


FIGURE 1: IMPOSED FLOW ANGLE AND DEVIATION AT DIFFERENT FLOW REGIMES (LEFT) AND MODIFIED CURVATURE DISTRIBUTIONS EXTENDING 10% 30% AND 60% IN THE STREAMWISE DIRECTION FROM THE TE. THE FLOW IS ENTERING FROM THE TE (REVERSE FLOW).

of the two streams primarily occurs downstream (outside) of the blade passage. Both assumptions align with the experimental observations of Gamache and Greitzer [28], although the latter is less accurate for longer blades. The derivation of the model follows a similar approach to Longley's simplified blockage-mixing model [15], but omits simplifications unsuitable for impellers and vaned diffusers, such as constant inlet/outlet blade angles, areas, and radii. The bulk flow is considered to be perfectly aligned with the camberline and fully isentropic; flow deviation and momentum transfer due to shear layers is considered negligible. To facilitate the derivation of the model, additional assumptions are made; the flow is considered incompressible and the separation bubble invariable and uniform, i.e. $\overline{b_{aero}} = \text{const}$. Compressibility effects are not expected to significantly affect the model's accuracy. However, the constant blockage assumption overlooks partial mixing inside longer blade passages, which may lead to an overestimation of mixing losses and deviations in the predicted flow direction. The assumption of negligible deviation may become inaccurate in the presence of strong gradients, reducing the accuracy of the predicted mixed-out flow angle. The assumption of strictly axial or radial flow is reasonable for most aero-engine turbomachinery but may be limiting for other applications. A schematic representation of the assumed passage flow structures and inlet/outlet geometrical and flow conditions are depicted in Fig. 2.

Blockage model. A control volume analysis is carried out, schematically shown in Fig. 2, the boundaries of which, coincide with the ones of the flow inlet (1) and outlet (2) of the blade passage as well as the location of the fully mixed-out flow (3). Variations in inlet/outlet blade angles, areas and radii are considered in the formulation of the model. Combining continuity, momentum conservation in the streamwise direction (relative FoR) and conservation of rotary stagnation pressure in the bulk flow (assumed isentropic), the blockage parameter is derived in

the form of a quadratic function as per Eq. (7). More details on the derivations of the blockage-mixing model are given in the Appendix.

$$\alpha \overline{b_{aero}}^2 - 2\overline{b_{aero}} + 1 = 0 \quad (7)$$

,where the coefficient α is given by Eq. (8).

$$\alpha = \overline{b_{aero}}^{-2} \left\{ \left(\frac{A_2 \cos \kappa_2}{A_1 \cos \beta_1} \right)^2 \left[2 \frac{P_1}{\rho W_1^2} \left(\frac{A_1 \cos \kappa_1}{A_2 \cos \kappa_2} - 1 \right) - \frac{(U_2^2 - U_1^2)}{W_1^2} + 2 \left(\frac{A_1 \cos \beta_1 \cos (\beta_1 - \kappa_1)}{A_2 \cos \kappa_2} \right) + \frac{2}{A_2 \cos \kappa_2 W_1^2} \iiint f_{cyl} \hat{dV} - 1 \right] \right\} \quad (8)$$

The volume integral in Eq. (8) represents the source terms which arise due to the formulation of the governing equations in curvilinear coordinates. These can be determined analytically as per Eq. (9), by setting the aerodynamic blockage parameter to unity, for a zero incidence angle, i.e. $\overline{b_{aero}} = 1$ and $\beta_1 = \kappa_1$. This ensures that no aerodynamic blockage will be imposed when the incidence angle at the TE is zero.

$$\iiint f_{cyl} \hat{dV} = \frac{\cos \kappa_2 A_2 W_1^2}{2} \left\{ \left(\frac{\cos \kappa_1 A_1}{\cos \kappa_2 A_2} - 1 \right)^2 + \frac{U_2^2 - U_1^2}{W_1^2} - 2 \frac{P_1}{\rho W_1^2} \left(\frac{A_1 \cos \kappa_1}{A_2 \cos \kappa_2} - 1 \right) \right\} \quad (9)$$

Constant-area (axial) mixing. During reverse flow operation, the mixing downstream the LE of impellers and axial compressor blades can be considered as a constant-area mixing process, since the flow is predominantly axial and the area

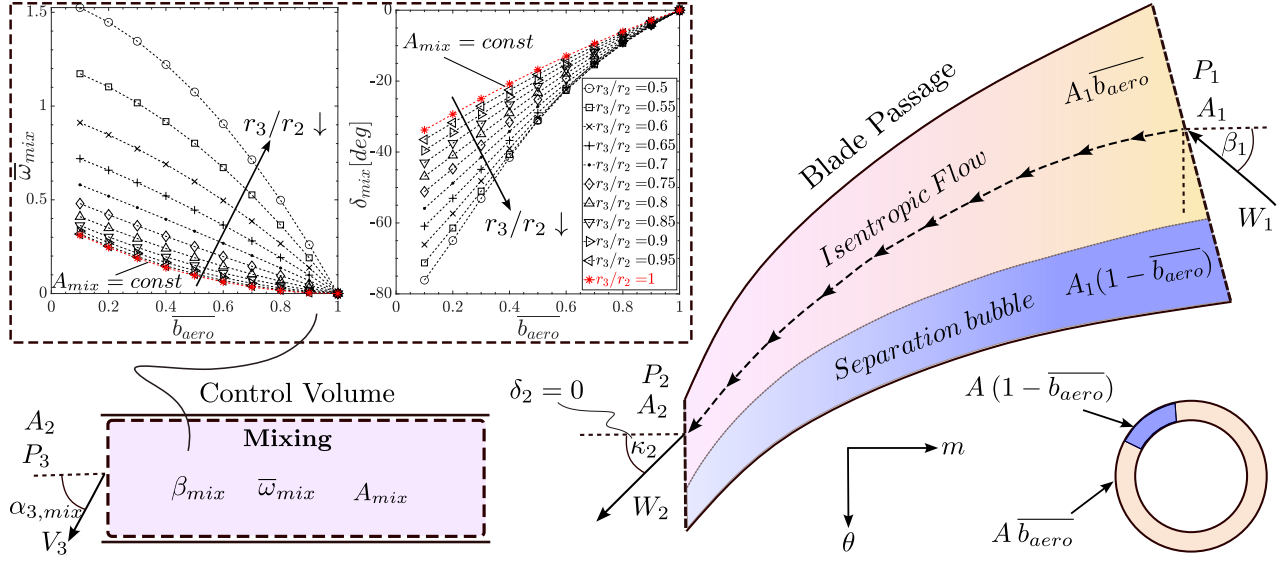


FIGURE 2: SCHEMATIC OF THE SIMPLIFIED BLADE-DUCT CONFIGURATION ILLUSTRATING THE ASSUMPTIONS OF THE BLOCKAGE-MIXING MODEL. THE CONSTANT-AREA DUCT USED FOR THE AXIAL MIXING MODEL IS SHOWN.

variations insignificant. The process along with the aforementioned assumptions for the derivation of the model are shown in Fig. 2.

The mixed-out flow angle is derived by combining continuity and circumferential momentum conservation in a control volume between the blade LE and the arbitrary location where the flow is fully mixed out. The flow angle at the LE is considered to be equal to the blade metal angle ($\delta_{LE} = 0$) and the rotational speed of the ducted domain zero. The mixed-out flow angle is then estimated by:

$$\alpha_{3,mix} = \text{atan} \left(\frac{\tan \kappa_2}{\overline{b_{aero}}} \right) \quad (10)$$

The loss coefficient due to mixing-out of the bulk flow and the separated wake is obtained from Eq.(11) by combining continuity and meridional momentum at the same control volume:

$$\overline{\omega}_{mix} = \cos \kappa_2^2 \left[\overline{b_{aero}}^2 \left(2 - \frac{1}{\cos \alpha_{3,mix}^2} \right) - 2\overline{b_{aero}} + \frac{1}{\cos \kappa_2^2} \right] \quad (11)$$

,where the loss coefficient is defined with respect to the passage outflow dynamic head at the LE as:

$$\overline{\omega}_{mix} = \frac{P_{03} - P_{02}}{\frac{1}{2} \rho V_2^2} \quad (12)$$

Variable area (radial) mixing. A similar control volume analysis is carried out considering a strictly radial geometry, i.e. $\varphi = 90^\circ$, whereby both the inlet and outlet areas are purely a function of radius. This enables applicability of the model to any radially positioned blade passage, e.g. vaned diffusers (mixing in vaneless space), radial impellers etc. Centrifugal compressors typically have axial inlets, thus the axial mixing model is used instead. Unlike constant area mixing, in radial

mixing the mixed-out state is dependent on the mixing rate and in turn, the control volume's extent. The radial boundaries of the control volume also define the inflow/outflow areas which vary significantly, designating the radial position where the flow is expected to be fully mixed-out. Using a free vortex approach [29], i.e. $V_\theta \sim 1/r$, combining continuity and conservation of circumferential momentum, the change in flow direction due to mixing-out of the flow is estimated by Eq. (13):

$$\alpha_{3,mix} = \text{atan} \left[\tan \kappa_2 \frac{r_3}{r_2} \left(\frac{1}{\overline{b_{aero}}} - \frac{4}{\left(1 + \frac{r_3}{r_2}\right)^2} \left(1 - \frac{r_3}{r_2}\right) \right) \right] \quad (13)$$

The loss coefficient due to mixing-out of the flow, is derived by combining continuity, meridional momentum and energy:

$$\begin{aligned} \overline{\omega}_{mix} = & \frac{P_2 \left(1 - \frac{r_2}{r_3}\right)}{\frac{1}{2} \rho V_2^2} + \frac{r_2}{r_3} \cos \kappa_2^2 \left\{ \overline{b_{aero}}^2 \left[\frac{r_3}{r_2} \left(2 - \frac{1}{\cos \alpha_{3,mix}^2} \right) \right. \right. \\ & \left. \left. - 8 \tan \kappa_2^2 \frac{\left(1 - \frac{r_3}{r_2}\right)}{\left(1 + \frac{r_3}{r_2}\right)^2} \right] - 2\overline{b_{aero}} + \frac{r_3}{r_2} \frac{1}{\cos \kappa_2^2} \right\} \quad (14) \end{aligned}$$

It is important to note, that due to the free vortex formulation, $\delta_{3,mix} \neq 0$ and $\overline{\omega}_{2-3} \neq 0$ even if $b_{aero} = 1$, thus the actual loss and flow deviation is the difference between the zero incidence and the computed ones. In variable area mixing, the flow direction and the losses are a function of one additional variable, i.e. r_3/r_2 , the definition of which, is rather arbitrary. It is therefore essential to identify its contribution relative to that of the blockage

parameter. The loss coefficient and the flow deviation are plotted in a range of blockage parameters and radius ratios, shown in Fig. 2. Close to 0.9 radius ratio range, representative of vaneless space radius ratio, the loss coefficient is primarily governed by the aerodynamic blockage rather than the radius ratio. At lower r_3/r_2 , the loss coefficient is governed by both the aerodynamic blockage and the radius ratio. Focusing on the deviation angle, it is evident that both the radius ratio and the aerodynamic blockage have a noteworthy effect throughout the entire range, with the effect of the latter being more dominant in the high $\bar{b}_{aero}$ region. For the limiting case where $r_3/r_2 = 1$, the variable area mixing loss and deviation equations reduce to the constant area ones, shown in red.

3. PLENUM AND THROTTLE MODEL

In experimental compressor rigs, the physical outlet of the compressor is typically attached to a large volume tank (plenum), the pressurisation of which, is controlled via the opening or closing of the throttle downstream the plenum. The numerical modelling of a throttle-plenum configuration could be performed either explicitly, via the generation of a discretised domain, representative of the real experimental plenum-throttle configuration or implicitly, via a dynamic change of the boundary conditions at the physical outlet of the compressor in accordance to a numerical, physics-based, 1D model.

The first, provides a more realistic modelling method enabling more precise capturing of both the direction and reflection of pressure waves inside the plenum. These are known to affect stall inception and the dynamics of the surge event [17]. In practice, this approach is often not feasible due to the complex ducting and other components, or in other occasions, these data are not available. The latter approach provides an efficient modelling alternative, whereby the entire plenum-throttle system is numerically represented via an 1D model. While geometry-related phenomena, e.g. reflection of pressure waves cannot be replicated, this approach provides extended flexibility since both the plenum volume and the throttle can be modified instantly and in a transient manner (throttle adjustment).

The integrated plenum-throttle model is similar to the one proposed by Righi [18] and is governed by Eqns. (15),(16), with subscripts Pl and T referring to plenum and throttle quantities respectively:

Continuity

$$\frac{\partial \rho_{Pl}}{\partial t} = \frac{1}{V_{Pl}} \left(\sum (\rho u_n A)_{outlet} + \dot{m}_T \right) \quad (15)$$

Energy

$$\frac{\partial \rho_{Pl} E_{Pl}}{\partial t} = \frac{1}{V_{Pl}} \left(\sum (\rho u_n A H_0)_{outlet} + \dot{m}_T C_p T_{0Pl,abs} \right) \quad (16)$$

The throttle is treated as a simple convergent nozzle, the throat area of which, A_T , regulates the mass flow capacity of the system and can be updated transiently. Isentropic relations are used to determine whether the nozzle is choked. The time derivative terms are solved using a 4th order Runge-Kutta method.

4. REVERSE FLOW MODELLING VERIFICATION

To assess the validity of the blockage-mixing model, the flow-field solution of NASA's High Efficiency Centrifugal Compressor (HECC) [30] vaned diffuser is obtained at reverse flow conditions via 3D RANS simulations using ASNYS CFX. It is preferred over the wedge-type CC3 diffuser, due to its aerofoil-type vane design being representative of modern, high efficiency vaned diffusers. To account for potential asymmetries in the flow-field extending beyond a blade passage pitch, three passages are modelled as illustrated in Fig. 3a. The domain is fully structured, with a total cell count of 1.24×10^6 elements. The average y^+ value along the blade surfaces is approximately 1.53. The $\kappa - \omega$ SST turbulence model is used, following the findings of [31] and its performance in highly separated turbomachinery flows. Higher-order schemes are used for the advection and turbulence terms. To capture the fully mixed-out flow, the domain is entirely radial and extended by approximately 10 radial chords (c_r). This required the translation of the geometry in the radial direction and adequate scaling to preserve the original blade pitch and blade geometry. Total pressure and temperature are imposed at the inlet along with a direction vector to adjust the inlet flow angle. Mass flow is imposed at the outlet. Pitchwise-averaged properties are extracted at several planes along the blade passage and the outlet domain, depicted in Fig. 3a. Inlet and outlet properties are then extracted at the corresponding blade passage and outlet domain planes to obtain the required inputs for the proposed model. Flow-field solutions are obtained for different incidence angles in the range $[30^\circ - 100^\circ]$.

4.1 Aerodynamic blockage

The distribution of the CFD-derived, average aerodynamic blockage along the blade passage's planes is plotted at different incidence angles in Fig. 3b. The effective area is defined as $A_{eff} = \dot{m} / \rho V_{mer}$ and the blockage as $b_{aero} = A_{eff} / A$, where A the available flow area with metal blockage. The maximum aerodynamic blockage, expressed by the minimum ($b_{aero} A$) value, manifests at approximately 20% of the radial chord, downstream of which, it progressively dissipates, reaching its minimum at the LE. The size of the separation bubble gradually diminishes as the flow travels towards the LE, as shown by the total pressure contours in Fig. 3d,e. This effect is not as pronounced in axial compressor blades, as the substantial reduction in the area from the TE to the LE accelerates the flow, enhancing mixing and accelerating the suppression of the separation bubble. As expected, the incidence angle is the predominant flow property, governing the aerodynamic blockage, which exceeds 50% at $i = 81^\circ$.

Since blockage varies across the passage, the mean aerodynamic blockage is estimated by averaging the blockage extracted from the CFD model at multiple passage planes. This is subsequently compared against the analytical model predictions in Fig. 4, obtained using Eqns. (7), (8). The model predictions express the average passage blockage (assumes $\bar{b}_{aero} = \text{const.}$) and are dependent only on inflow conditions, obtained at the plane positioned at r_1 shown in the same figure. The predictions of the model are in very good agreement with the CFD ones, which tends to slightly underestimate the blockage in the lower incidence range and slightly overpredict it in the higher incidence

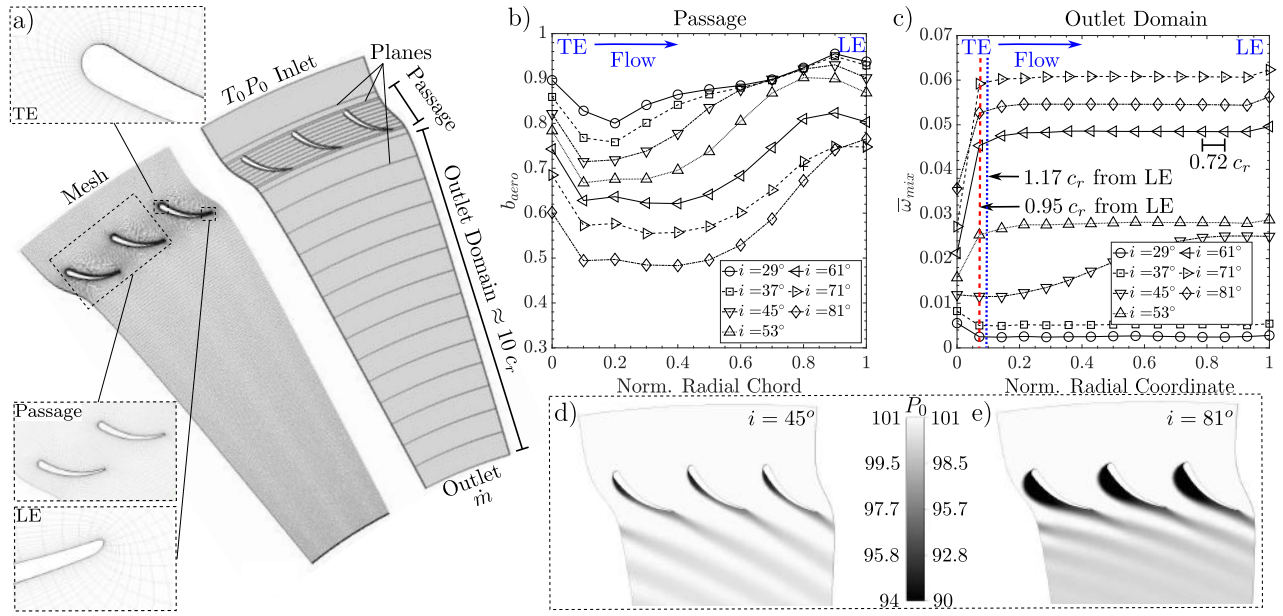


FIGURE 3: A) NASA HECC VANED DIFFUSER COMPUTATIONAL DOMAIN, B) PASSAGE AERODYNAMIC BLOCKAGE, C) MIXING LOSS COEFFICIENT AT THE OUTLET DOMAIN AND TOTAL PRESSURE CONTOURS AT INCIDENCE ANGLE OF D) 45° AND E) 81°.

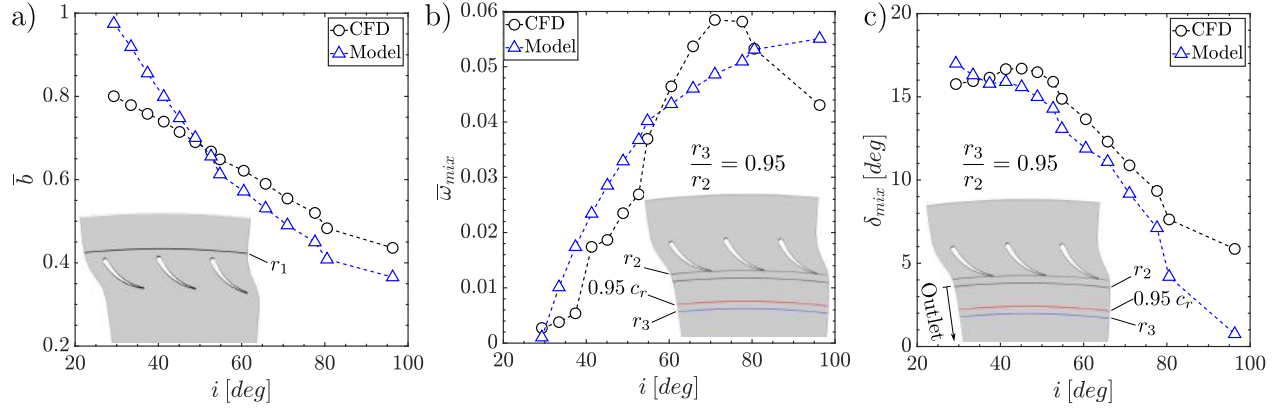


FIGURE 4: COMPARISON OF ANALYTICAL AND CFD-DERIVED A) AERODYNAMIC BLOCKAGE IN THE BLADE PASSAGE, B) MIXING LOSS COEFFICIENT AND C) DEVIATION ANGLE DUE TO MIXING AT THE OUTLET DOMAIN.

range. In the $[50^\circ - 90^\circ]$ range, the trends between the two lines are consistent and in good agreement.

4.2 Mixing losses

A similar analysis is carried out for the verification of the mixing loss model. The total pressure losses associated with the mixing process of the separated wake and main flow are quantified with the mixing loss coefficient, $\bar{\omega}_{mix}$, defined using the local total pressure (at each plane) as well as total pressure and dynamic head at the first plane of the outlet domain (flow inlet plane of outlet domain). As the flow travels backwards, the acceleration of the flow drastically reduces the mixing time of the wake and the main flow. This is evident by the $\bar{\omega}_{mix}$ curves (Fig. 3c), which flatten out at just 0.95 radial chords downstream of the LE. The predictions of the analytical model are subsequently compared against the CFD-extracted values. To ensure consistency with the model, all inlet/reference properties are computed at the plane just downstream of the LE, corresponding to r_2 (see Fig. 4b). The

radius ratio, r_2/r_3 , which yields the most accurate predictions is 0.94-0.95, located $1.17 c_r$ downstream of the LE, as shown in Fig. 4b). This is reasonably close to the location where most of the mixing takes place ($\approx 0.95 c_r$). The predictions of the analytical model are in very good agreement with the CFD data across the entire incidence range with the most notable discrepancy occurring at the highest incidence angle.

4.3 Mixed-out flow angle

The model input properties and the CFD-derived deviation angle are extracted at the same planes. The mixed-out flow angle is obtained by subtracting the flow angle at the LE from the mixed-out/predicted one. A comparison between the CFD data and the analytical model are shown in Fig. 4c. The model accurately predicts the change in the flow direction across the entire range, slightly underpredicting it, except for the highest incidence point where a higher discrepancy is recorded.

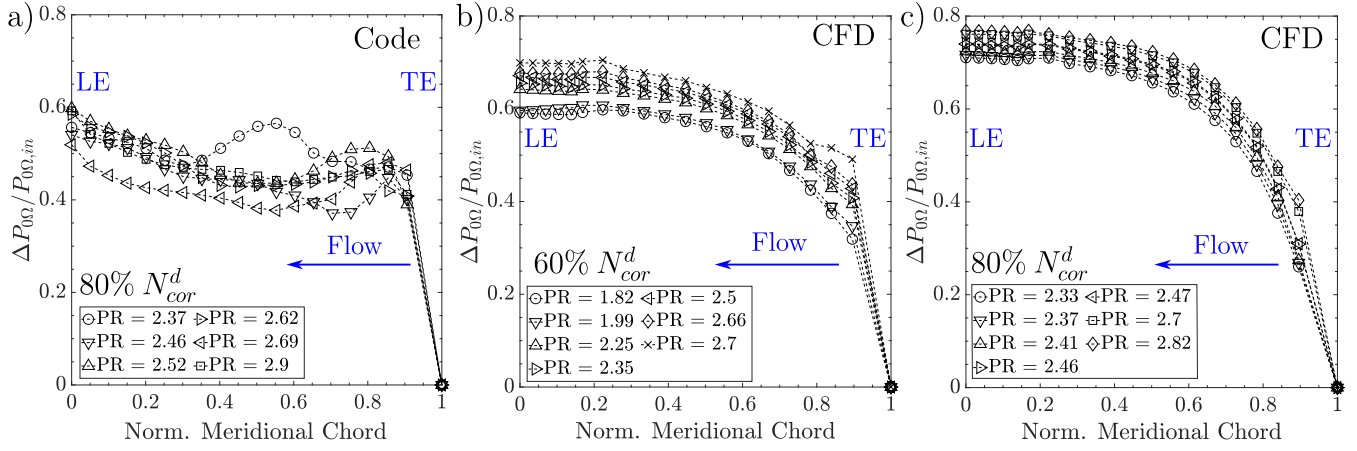


FIGURE 5: NORMALISED ROTARY STAGNATION PRESSURE LOSS ALONG CC3 IMPELLER PASSAGE FOR DIFFERENT PR EXTRACTED FROM A) THROUGH FLOW ($80\%N_d$), B) CFD ($60\%N_d$) AND C) ($80\%N_d$) CFD SIMULATIONS.

4.4 Separation losses

Figure 5a shows the normalised rotary stagnation pressure loss along the impeller CC3 passage at 80% speed and reverse flow conditions from steady-state, through-flow simulations and pressure ratios. The reattachment point is set to 10% meridional chord. The corresponding distributions, obtained from 3D RANS simulations and pitchwise-averaged flow-fields at 60% and 80% are shown in Figs 5b, c. Open boundary conditions (P_0, T_0) are imposed at the flow inlet and outlet. The outflow pressure is set to 1 atm, the inflow total pressure is varied to obtain different pressure ratios, while the imposed total temperature is kept at 288K. The through-flow loss distributions are in very good agreement with the 60% speed curves although consistently lower compared to the 80% speed curves. The abrupt jump in pressure losses in the vicinity of the TE due to flow separation, is not only replicated by the code in a physically accurate manner, but its magnitude and chordwise position correlate exceptionally well with the CFD data. As the flow convects towards the LE, the losses progressively increase with a continuously decreasing gradient, which approaches zero close to the LE. This is due to the combined contribution of skin friction, gradual reduction of the separation bubble and the mixing of the separated swirling flow. Despite the complexity of these flow features, the proposed modelling approach reproduces the passage loss distribution with reasonable accuracy. The associated losses are considered by introducing an additional skin friction loss in the passage domain.

5. NASA CC3 POST-STALL AERODYNAMICS

The NASA CC3 compressor is modelled to validate the methods proposed herein. It is the only centrifugal compressor found in the public domain with both geometrical and experimental surge data available. It essentially is an up-scaled Alison compressor with a design mass-flow approximately three times higher than the original design (4.54 kg/s instead of 1.658 kg/s) and a design pressure ratio of 4:1 [25]. The design corrected speed is $N_{cor}^d = 21789$ rpm. The impeller consists of 15 + 15 main and splitter blades with cut-off trailing edge design, while the vaned diffuser comprises 24, wedge-type blades. A 3D visualization of the full compressor geometry performed within AXSTAR is

shown in Fig. 6a.

Through-flow simulations are carried out on the meridional grid depicted in Fig. 6a. The spatial curvature and thickness distributions are estimated directly from discretised pressure, suction and camber surfaces of both blades. Contours of the computed blade angle, lean angle and blockage parameter distributions on the meridional plane are also shown in the same figure (bottom). The computational grid is generated using the adaptive mesh approach with a total element count of 7740. The grid consists of 20 spanwise elements, while the impeller and the vaned diffuser domains are discretised using 160 and 96 elements in the streamwise direction, respectively. It should be noted, that this grid resolution is substantially higher, compared to the recommended range [24], but a more detailed flow-field resolution was sought, as experimental DPIV measurements are available. For simplicity, the pressure losses and the deviation angle are distributed uniformly in the spanwise, and linearly in the streamwise direction (forward and reverse flow), since they are not expected to introduce any noteworthy errors in global aerodynamic performance metrics [24]. The reattachment point is set to 10% of the meridional chord.

Due to the absence of experimental, steady-state, reverse flow performance data, reverse flow characteristics at 40%, 60%, 80% and 100% corrected design speed are initially obtained through 3D, single-passage, RANS simulations. Since surge and by extension, reverse flow operation primarily exhibit axisymmetric behaviour [18], single passage simulations are deemed appropriate. Following the study of [32], a structured grid of 12 million elements was generated, shown in Fig. 7b. The grid was generated in ANSYS Turbogrid, and $\kappa - \omega$ SST turbulence model was used, in accordance to the findings of [31] and previous experience in highly separated turbomachinery flows. The solution of the flow-field is performed in CFX using higher-order accurate schemes for the advection and turbulence terms. The impeller and vaned diffuser domains are numerically adjoined via a mixing plane. The model has been validated at forward flow conditions against the experimental data of Skoch et al. [33].

A comparison of the pressure and torque characteristics obtained from through-flow and 3D RANS simulations are plotted

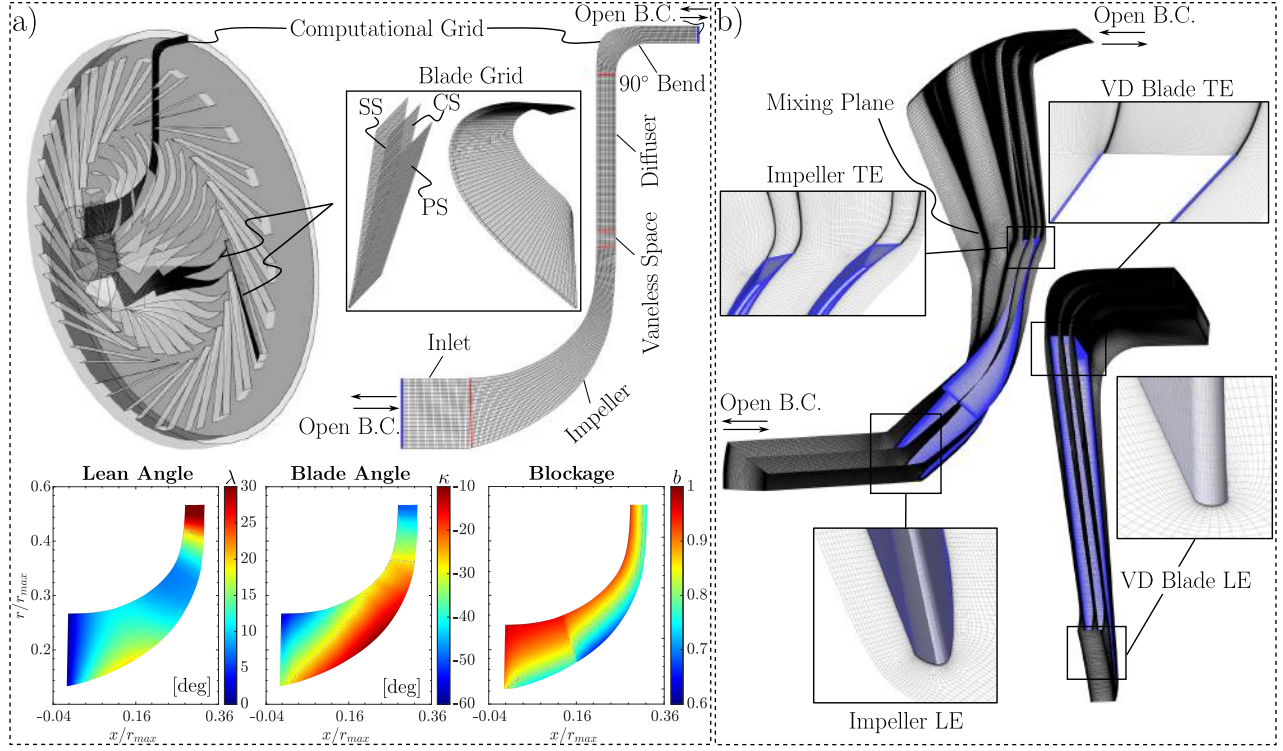


FIGURE 6: A) 3D VISUALISATION OF NASA CC3 COMPRESSOR, BLADE GRIDS FOR CURVATURE AND BLOCKAGE DEFINITION, THROUGH-FLOW GRID, BLOCKAGE AND BLADE ANGLE DISTRIBUTIONS AND B) RANS COMPUTATIONAL GRID.

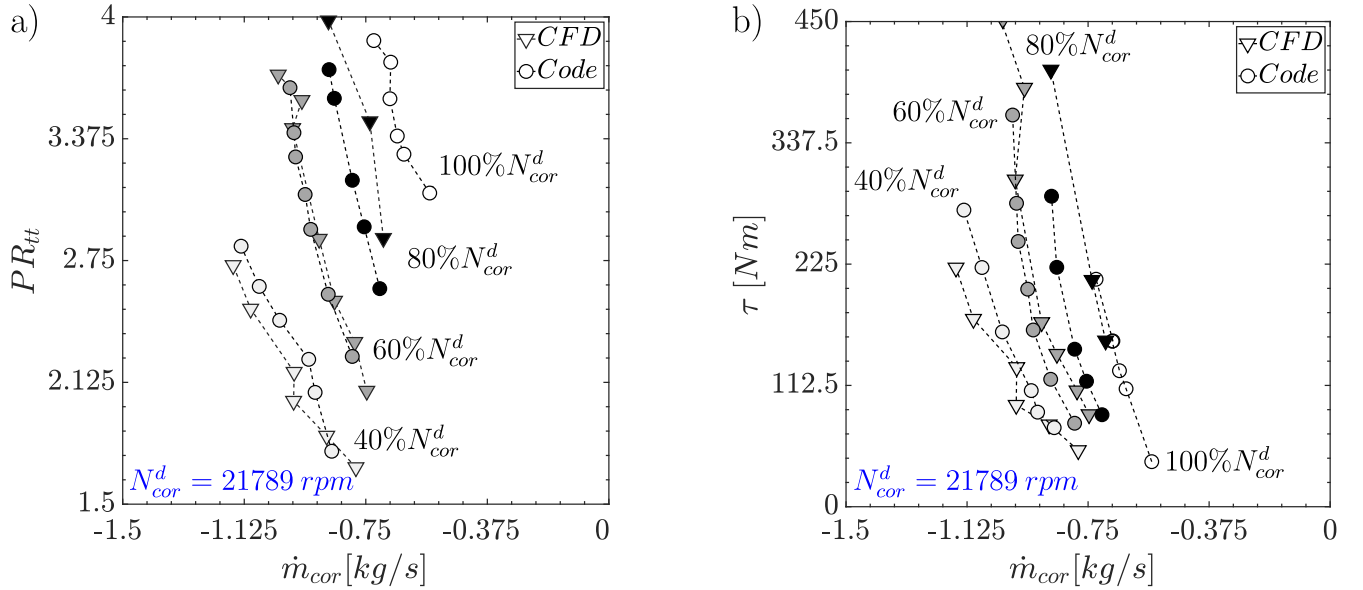


FIGURE 7: COMPARISON OF A) TOTAL-TO-TOTAL PRESSURE RATIO AND B) TORQUE REVERSE FLOW CHARACTERISTICS OF COMPRESSOR CC3 WITH VANED DIFFUSER OBTAINED WITH AXSTAR AND RANS SIMULATIONS.

in Fig. 7. The mass flow and the corrected properties are computed using inlet flow conditions (at the geometrical outlet). The pressure ratio is defined as in forward flow, i.e., the ratio of total pressure at the geometric outlet to that at the inlet. The derivation of reliable, steady-state operating points at 100% design speed was particularly challenging due to the large oscillations in the mass flow and pressure ratio and are not included. Similar is-

sues were encountered at 80% speed, nevertheless a few stable operating points were extracted and included in the dataset. The total-to-total pressure characteristics obtained with the code are in very good agreement with the CFD results along the entire speed range both in terms of mass-flow capacity and pressure rise. At 40% speed, the pressure rise is slightly overestimated by the code, while at 80% speed, the differences are slightly larger

and primarily linked either to blockage and/or underestimation of the losses, as discussed earlier (see Fig. 5). The torque predictions also correlate exceptionally well with the CFD predictions in the lower 40% and 60% speed range, and exhibit a similar trend at 80% speed, being slightly shifted towards the lower mass flow and torque range. The steady-state results indicate that the primary modelling aspects e.g. flow turning, losses, blockage and the energy exchange are adequately modelled and predicted in the through-flow code, across a range of mass flows and speeds.

To model the transient, post-stall aerodynamic response of the CC3 compressor, a plenum is required to replicate the experimental configuration and capture the volume dynamics of the system. In the experimental configuration [26], an extended inlet is attached in front of the impeller, while the presence of ducting components and a plenum volume are reported. Geometric or sizing data however are not provided. The reported frequency of the surge cycles is approximately 3Hz and the compressor operates at 100% speed. The experimental data comprise pressure measurements obtained using fast-response, dynamic pressure transducers during surge events at three locations; vaneless space, vaned diffuser throat and impeller inlet. DPIV methods are also carried out, enabling visualisation of the flow-field at the critical, from a stability point of view, vaned diffuser passage.

Due to the absence of sizing and geometrical data regarding the experimental configuration's inlet, plenum and intermediate ducting components, the initial through-flow computational domain is kept unchanged. The pressure outlet is removed and dynamically varied according to the state of the 1D plenum-throttle model discussed earlier. The onset of the surge event, was performed by entirely closing the plenum's throttle, while the compressor was operating at 100% speed and a stable, low mass flow setting. Once the compressor surged, the throttle was opened to enable recovery. The volume of the plenum was varied until the frequency of the surge cycles matched the experimentally reported one. This resulted in a plenum volume of about $V_p = 0.25m^3$, during which, 4 cycles were recorded. The duration of each surge cycle is about 0.33 seconds, translating to approximately 121 revolutions per surge cycle. The throttle was kept 80% open throughout the entire surge event due to the compressor's tendency to recover after the completion of the first surge cycle when the throttle was left fully open.

Figure 8 shows the transient pressure recordings obtained with through-flow simulations at the impeller inlet and the vaned diffuser throat for 2 plenum volume sizes: $0.15m^3$ and $0.25m^3$. As expected, the larger volume (Fig. 8a) exhibits lower surge frequency with slightly higher pressure peaks than the lower volume configuration. During both simulations, the throttle was 80% open, precipitating 3 and 4 surge cycles respectively, before the compressor recovered. The pressure traces at the inlet as well as the magnitude and the shape of the cycle pressure signals are consistent with the experimental recordings. The two surge cycles exhibit different patterns primarily in the region preceding the pressure peak. During the recovery phase of the first cycle, oscillations of increasing amplitude manifest. Their amplitude increases rapidly but the disturbances gradually dissipate. Their origin is discussed later in the section. Shortly after the dissipation of the disturbances, a spike-type trace is shown, a few revolu-

tions after which, the surge event initiates. This spike precedes all surge cycles of that particular surge event. Spike stall inception normally initiates from a single-passage instability which rapidly propagates in the circumference. The classification of that phenomenon as spike-stall inception could be deemed ambiguous, due to the axisymmetric nature of the solver. It is hypothesised that spike stall inception could manifest in the code, since spikes rapidly propagate around the circumference, growing into a local, axisymmetric disturbance shortly after their emergence in a blade passage.

The second surge event (Fig. 8b), evinces a different stall inception pattern, resembling a modal wave-type of stall inception. Perturbations of increasing amplitude emerge during the re-pressurisation phase. These are not damped, but rather grow progressively and more rapidly a few revolutions prior to the onset of each surge cycle. As modal waves are primarily a circumferentially-independent type of instability, driven by the impeller-diffusion interaction [34], their manifestation within the current through-flow framework is deemed physically possible. The stall inception mode, i.e. spike or modal, is not reported in the experimental study [26], while no conclusive result can be made by inspection of the data to identify the validity of the stall inception mechanism predicted herein.

To validate the simulated 3Hz surge cycles, the recorded pressure recordings at the vaned diffuser inlet are compared against the corresponding experimental ones. The simulation of the entire surge event was completed in less than 6 hours, running on a single core of a regular desktop computer's processor. Figure 9 shows a comparison between the experimental and simulated static pressure measurements at the throat of the vaned diffuser along with streamwise velocity contours at different time instances. The simulated pressure traces shown in Fig. 9 are in excellent agreement with the experimental results. The magnitude of the highest (1) and lowest (2) pressure peaks are accurately predicted by the through-flow code. Once peak pressure is reached, the entire compressor stalls, with fully reverse flow developing in the duct (1). The lowest pressure point (2) corresponds to a state where the pressure gradient driving the reverse flow has reached equilibrium with the pressure rise developed by the impeller. This allows forward flow to re-establish across extensive regions of the impeller and diffuser. The most noteworthy discrepancy between the experimental and simulated cycles is the duration of the region between the highest and lowest pressure peaks, the duration of which, is slightly overestimated by the code. An instability emerges and subsequently dissipates during the recovery phase which is not easily discernible in the experimental dataset, without affecting the dynamic behaviour of the surge cycles. This originates from a pressure-driven interaction between the stalled flow in the vaneless space and the 90° exit duct, near the shroud (3). This persists, as the low pressure is insufficient to clear them away (3). As the pressure increases, this interaction causes stalling (4) and unstalling (5) of the diffuser section in a cyclic manner, until the pressure is sufficiently high to remove them. The spike in (6) seems to be associated with a pressure wave emanating near the vaneless space shroud, suppressing the stalled flow in the diffuser shroud.

Figure 10 shows the total-to-total pressure ratio and torque

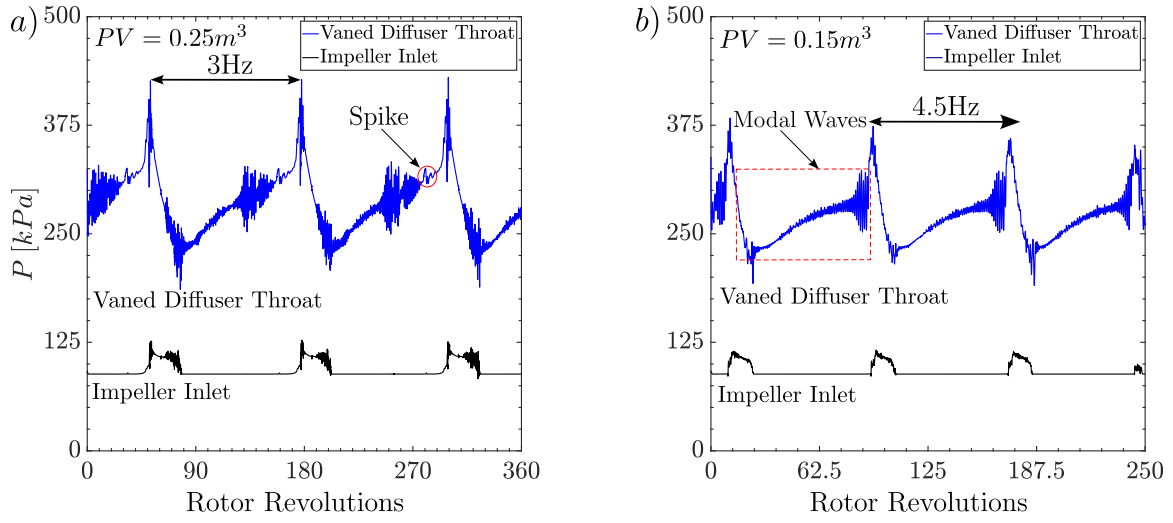


FIGURE 8: NASA CC3 THROUGH-FLOW PRESSURE RECORDINGS AT VANED DIFFUSER THROAT AND IMPELLER INLET DURING A SURGE EVENT OBTAINED WITH A PLENUM VOLUME OF A) $0.25m^3$ AND B) $0.15m^3$.

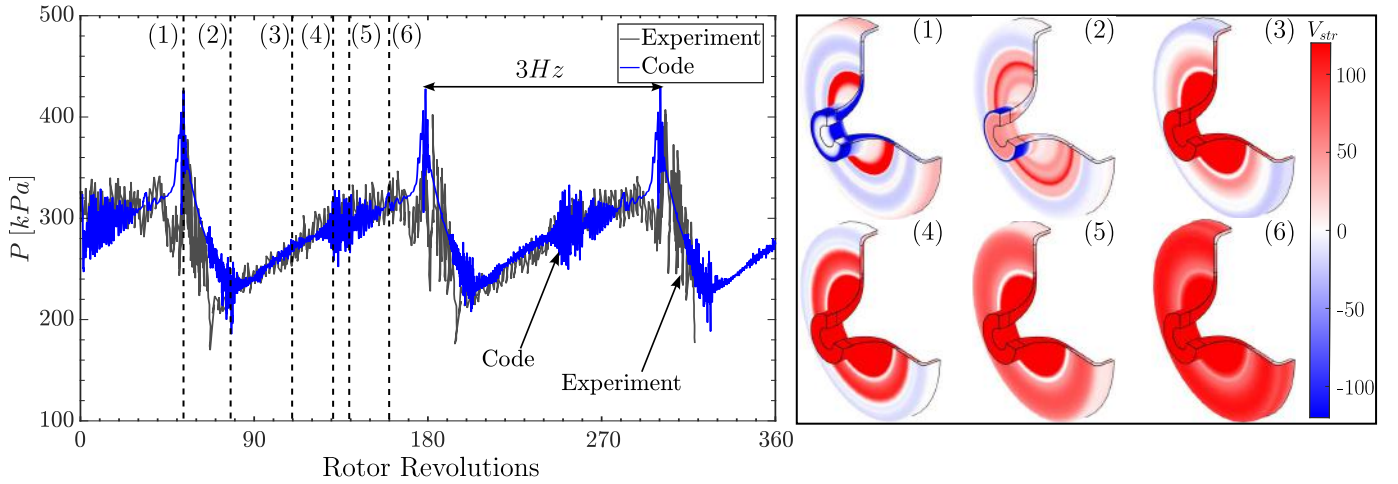


FIGURE 9: STREAMWISE VELOCITY CONTOURS AND COMPARISON OF NASA CC3 THROUGH-FLOW PRESSURE RECORDINGS AT VANED DIFFUSER THROAT AGAINST EXPERIMENTAL MEASUREMENTS DURING A THREE-CYCLE SURGE EVENT WITH A $0.25m^3$ PLENUM.

transient measurements during the first 3Hz surge cycle simulated with AXCSTAR (see Fig. 9). The $100\%N_d$ forward flow along with the 40%, 60% and 80% N_d characteristics obtained from steady-state 3D RANS simulations are also included along with the surge cycle trajectory. The mass flow is measured at the geometrical inlet and is not corrected as the principal flow direction changes during surge. The closed throttle pressurises the plenum which in turn, shifts the compressor's operating point towards the zero mass-flow region. The gradient of the characteristic changes and preserves a near-zero gradient; ideally, proper modelling would require full wheel simulations as rotating stall phenomena might manifest. In centrifugal compressors, these are known to only mildly penalise the pressurisation in the positive, low flow region [4], which along with the rapid transient response, are not expected to introduce substantial errors in the predicted response. The compressor exhibits deep surge with the cycle trajectory being aligned with the 80% reverse flow characteristic. During flow reversal, the total temperature in the plenum

ranges between [410K - 490K] translating to a corrected speed within a [0.77% - 0.83%] range, which is in agreement with the CFD-extracted 80% reverse flow characteristic.

In the positive, zero-gradient region of the pressure ratio characteristic, the torque progressively drops and decreases abruptly as it approaches zero mass-flow. In the highest negative mass flow region, the torque reaches its reverse flow peak value and keeps decreasing as it approaches zero mass flow. The torque seems to be better aligned with the 60% characteristic, although all torque characteristics are closely clustered together. Eight points corresponding to different time instances of the surge cycle are selected to visualise the flow-field.

The streamwise velocity contours at different instances during the surge cycle are illustrated in Fig 11. Snapshot (1) is captured in the zero-gradient region of the PR_{II} characteristic, where a lower-momentum flow develops in the 90° bend. Further increase in the static pressure of the plenum causes the diffuser to stall, developing flow-reversal at the vaneless space and the

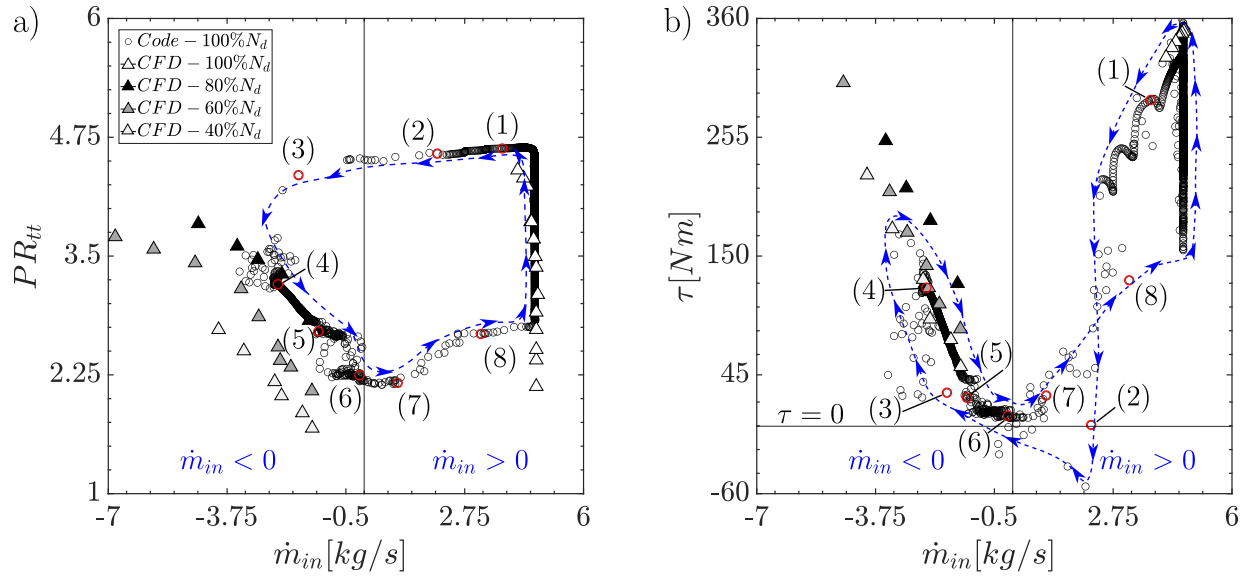


FIGURE 10: TOTAL-TO-TOTAL PRESSURE RATIO AND TORQUE DURING A SURGE CYCLE OBTAINED WITH THROUGH-FLOW SIMULATIONS AND A PLENUM VOLUME OF $0.25m^3$. STEADY-STATE CFD POINTS ARE ALSO INCLUDED IN THE GRAPH.

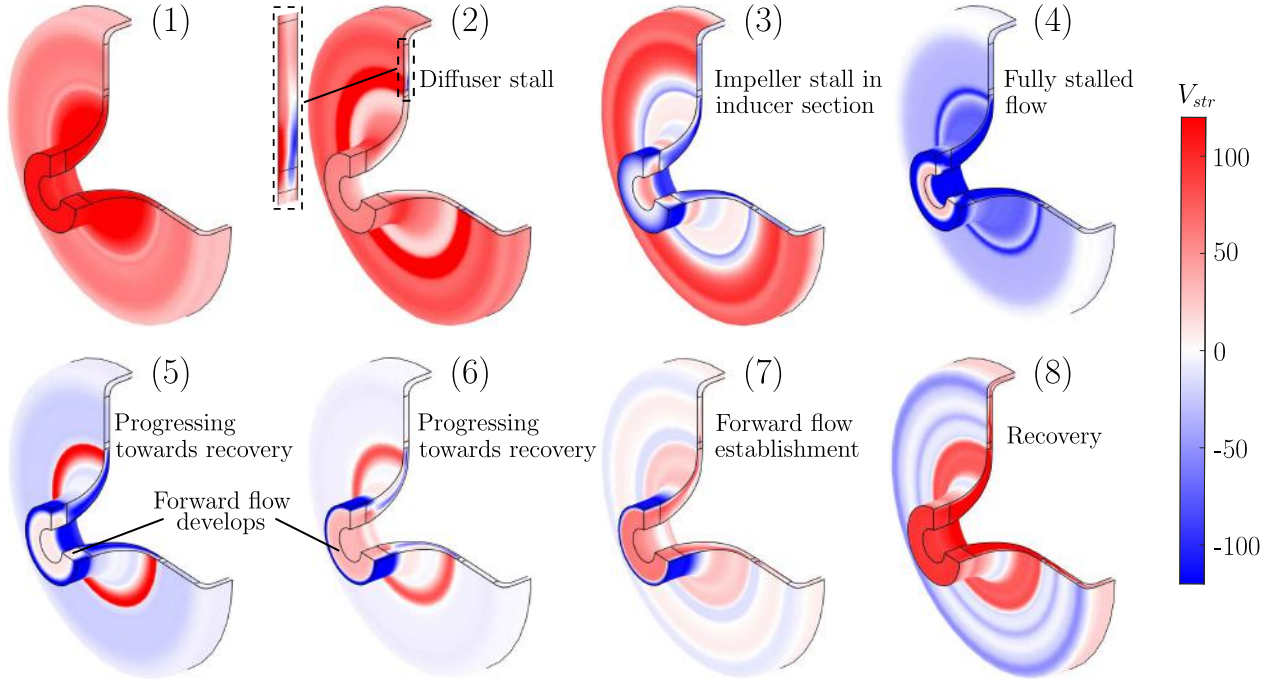


FIGURE 11: STREAMWISE VELOCITY CONTOURS AT DIFFERENT OPERATING CONDITIONS DURING A DEEP SURGE CYCLE OF NASA CC3 COMPRESSOR WITH VANED DIFFUSER WITH A PLENUM VOLUME OF $0.25m^3$.

vicinity of the vaned diffuser inlet and throat. This is in excellent agreement with Wernet et al. [26], who reported the onset of flow reversal at the exact same location using DPIV measurements. In the low, negative mass-flow region (3), the impeller stalls, particularly in the inducer section with flow reversal gradually developing along the entire duct except for a small flow stream near the shroud and a part of the blade passage near the impeller TE. A few instances later, fully reverse flow conditions establish along the entire duct (4). As the plenum further depressurises, the compressor progresses towards recovery. The

forward flow region at the inlet increases, impeding the outflow of reverse flow stream from the impeller passage at the LE. This continuously suppresses the reverse flow stream forcing it to exit from a small area near the impeller eye (5),(6). Forward flow conditions subsequently establish along the duct, with low momentum flow persisting in the diffuser and oscillating flow near the shroud (7). The low momentum fluid in the diffuser is displaced and only localised reverse flow remnants at the shroud are observed (8).

Filtered pressure recordings at the same location during an-

other surge event are shown in Fig. 12. Details regarding the frequency of the surge cycle are not reported, neither the conditions under which these were obtained, but both the values of the highest and lowest peaks are lower than the main ones shown in Fig. 9. Low-pass filtered pressure profiles during a surge cycle obtained with a plenum volume of 0.19m^3 and 0.25m^3 are compared against the experimental results. Both simulated profiles correlate very well with the experimental ones. The profile corresponding to the smaller plenum volume, more precisely approximates the experimental traces, accurately estimating both the magnitude of the pressure peaks and the duration between them. The large oscillations preceding the 0.19m^3 cycle are not evident in the experimental results although they could have been filtered out. The 0.19m^3 cycle exhibits a similar stall precursor pattern to that shown in Fig. 8b. The large amplitude oscillations prior to peak pressure are caused by the same mechanism causing the instability which emerges and dissipates during the recovery of the 0.25m^3 cycle (instances (4) and (5) in Fig. 9). However, these are not dissipated, but rather progressively amplified, causing the vaned diffuser and 90° exit duct near the shroud to stall simultaneously, forming a continuous reverse flow stream that spans both regions. The profile corresponding to the higher plenum volume is wider, while in both cases, the pressure during the recovery phase is slightly overpredicted.

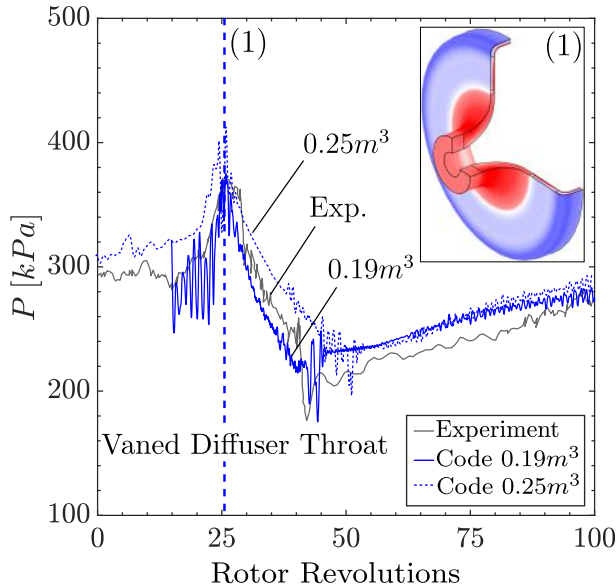


FIGURE 12: EXPERIMENTAL AND SIMULATED PRESSURE TRACES AT VANED DIFFUSER THROAT DURING A SURGE CYCLE WITH PLENUM VOLUME OF A) 0.19m^3 AND B) 0.25m^3 .

6. SUMMARY AND CONCLUSIONS

A complete, through-flow modelling approach is developed combining a custom solver in the relative FoR with metal and aerodynamic blockage and a custom body force model coupled with a modelling approach capable of dealing with uniform and highly stalled flows inside the passages. The latter blends standard empirical compressor correlations along with a new analytical blockage-mixing model and a body force-based approach to

replicate flow separation inside the passages. The formulation of the blockage-mixing model extends its applicability to axial blades, impellers and vaned diffusers. The reverse flow modelling approach is initially verified against 3D RANS data of the HECC vaned diffuser and the NASA CC3 impeller and finally the overall through-flow method is validated against experimental surge data of the NASA CC3 compressor.

The predictions of the blockage-mixing model are in very good agreement with the corresponding CFD-measured properties both in terms of magnitude and overall trend across the entire range of incidence angles, with slightly higher discrepancies recorded only at very high incidence angles. The imposed passage losses accurately reproduce the distribution of the losses inside the passage but tend to slightly underestimate the imposed losses. This is attributed partially to mixing and skin friction losses inside the passage. The NASA CC3 reverse flow characteristics obtained with through-flow simulations are in excellent agreement with the CFD data, particularly in the lower speed range. Deep surge cycles are subsequently obtained by varying the plenum volume size until the experimentally recorded cycle frequency is matched. The trajectory of the cycles are consistent with the steady-state, CFD forward and reverse flow characteristics, while the corrected rotational speed decreases during reverse flow operation due to changes in the plenum temperature. Spike and modal stall inception is captured when different plenum volume and throttle settings are imposed. The shape, frequency and magnitude of the experimentally recorded pressure traces at the diffuser throat are matched exceptionally well. The pressure traces at the impeller inlet are also successfully captured, while flow-field contours show the emergence of flow reversal in the vaneless space and the diffuser throat which is in agreement with the experimental DPIV measurements. The simulation of the entire surge event required approximately 6 hours on a regular computer, running on a single core. This translates to more than order of magnitude reduction in computational time compared to conventional CFD methods which typically would require multiple weeks on multi-core HPC systems.

ACKNOWLEDGMENTS

The authors would like to thank Rolls-Royce plc. for supporting this work and allowing its publication. Special thanks to Richard Tunstall, Steven Mazur and John Harvell for supervising this work.

REFERENCES

- [1] Kurzke, J., 2008, "Effects of inlet flow distortion on the performance of aircraft gas turbines," *J. Eng. Gas Turbines Power*, **130**(4), April 041201.
- [2] Kurz, R., and Brun, K., 2000, "Degradation in gas turbine systems," *J. Eng. Gas Turbines Power*, **123**(1), November, pp. 70–77.
- [3] Peitsch, D., 2000, "Modelling the transient behaviour of jet engines," In *Turbo Expo: Power for Land, Sea, and Air*, Vol. 1: Aircraft Engine; Marine; Turbomachinery; Micro-turbines and Small Turbomachinery V001T01A016.
- [4] Day, I. J., 2015, "Stall, surge, and 75 years of research," *J. Turbomach.*, **138**(1), October 011001.

- [5] Spakovszky, Z. S., and Roduner, C. H., 2009, "Spike and modal stall inception in an advanced turbocharger centrifugal compressor," *J. Turbomach.*, **131**(3), April 031012.
- [6] Fidalgo, V. J., Hall, C. A., and Colin, Y., 2012, "A Study of Fan-Distortion Interaction Within the NASA Rotor 67 Transonic Stage," *J. Turbomach.*, **134**(5), 05, p. 051011.
- [7] Zhao, F., Dodds, J., and Vahdati, M., 2018, "Poststall Behavior of a Multistage High Speed Compressor at Off-Design Conditions," *J. Turbomach.*, **140**(12), 10, p. 121002.
- [8] Shahin, I., Gadala, M., Alqaradawi, M., and Badr, O., 2015, "Large eddy simulation for a deep surge cycle in a high-speed centrifugal compressor with vaned diffuser," *J. Turbomach.*, **137**(10), June, p. 101007.
- [9] Gong, Y., 1999, "A Computational Model for Rotating Stall and Inlet Distortions in Multistage Compressors," Phd thesis, Massachusetts Institute of Technology.
- [10] Hsiao, E., Naimi, M., Lewis, J. P., Dalbey, K., Gong, Y., and Tan, C., 2001, "Actuator Duct Model of Turbomachinery Components for Powered-Nacelle Navier-Stokes Calculations," *J. Propul. Power*, **17**(4), pp. 919–927.
- [11] Chima, R. V., 2006, "A Three-Dimensional Unsteady CFD Model of Compressor Stability," In Proceedings of ASME Turbo Expo: Power for Land, Sea, and Air, Vol. 6, pp. 1157–1168.
- [12] Stewart, M. E. M., 1995, "Axisymmetric Aerodynamic Numerical Analysis of a Turbofan Engine," In Proceedings of ASME Turbo Expo: Power for Land, Sea, and Air, Vol. 2 V002T02A013.
- [13] Marble, F., 2017, *Aerodynamics of Turbines and Compressors* Princeton University Press, Princeton.
- [14] Escuret, J., and Garnier, V., 1994, "Numerical Simulations of Surge and Rotating-Stall in Multi-Stage Axial-Flow Compressors," In 30th Joint Propulsion Conference and Exhibit.
- [15] Longley, J. P., 2007, "Calculating Stall and Surge Transients," In Proceedings of ASME Turbo Expo: Power for Land, Sea, and Air, Vol. 6, pp. 125–136.
- [16] Righi, M., Pachidis, V., Könözy, L., and Pawsey, L., 2018, "Three-Dimensional Through-Flow Modelling of Axial Flow Compressor Rotating Stall and Surge," *Aerospace Science and Technology*, **78**, pp. 271–279.
- [17] Righi, M., Pachidis, V., Könözy, L., Giersch, T., and Schrape, S., 2022, "Experimental Validation of a Three-Dimensional Through-Flow Model for High-Speed Compressor Surge," *Aerospace Science and Technology*, **128**, p. 107775.
- [18] Righi, M., Pachidis, V., and Könözy, L., 2020, "On the Prediction of the Reverse Flow and Rotating Stall Characteristics of High-Speed Axial Compressors Using a Three-Dimensional Through-Flow Code," *Aerospace Science and Technology*, **99**, p. 105578.
- [19] Kottapalli, A. P., 2013, "Development of a body force model for centrifugal compressors," M.sc. thesis, Massachusetts Institute of Technology.
- [20] Benneke, B., 2009, "A methodology for centrifugal compressor stability prediction," M.sc. thesis, Massachusetts Institute of Technology.
- [21] Spakovszky, Z. S., 2004, "Backward traveling rotating stall waves in centrifugal compressors," *J. Turbomach.*, **126**(1), March, pp. 1–12.
- [22] Powers, K. H., Kennedy, I. J., Brace, C. J., Milewski, P. A., and Copeland, C. D., 2021, "Development and validation of a model for centrifugal compressors in reversed flow regimes," *J. Turbomach.*, **143**(10), May 101001.
- [23] Powers, K. H., Brace, C. J., Budd, C. J., Copeland, C. D., and Milewski, P. A., 2020, "Modeling Axisymmetric Centrifugal Compressor Characteristics From First Principles," *J. Turbomach.*, **142**(9), 08, p. 091010.
- [24] Lamprakis, D., Righi, M., and Pachidis, V., 2025, "A New Body Force-Based Modelling Approach for Turbomachinery Aerodynamics: Formulation, Verification and Implementation Aspects," In Proceedings of ASME Turbo Expo 2025 Paper No. GT2025-153775.
- [25] McKain, T. F., and Holbrook, G. J., 1997, Coordinates for a High Performance 4:1 Pressure Ratio Centrifugal Compressor Tech. Rep. NASA-CR-204134, NASA Technical Report, Detroit Diesel Allison, Indianapolis, IN, United States, July.
- [26] Wernet, M. P., Bright, M. M., and Skoch, G. J., 2000, "An Investigation of Surge in a High-Speed Centrifugal Compressor Using Digital PIV," *J. Turbomach.*, **123**(2), 01, pp. 418–428.
- [27] Aungier, R. H., 2000, *Centrifugal Compressors: A Strategy for Aerodynamic Design and Analysis* ASME Press, January.
- [28] Gamache, R. N., and Greitzer, E. M., 1990, "Reverse Flow in Multistage Axial Compressors," *Journal of Propulsion and Power*, **6**(4), pp. 461–473.
- [29] Main, A. J., Oldfield, M. L. G., Lock, G. D., and Jones, T. V., 1997, "Free Vortex Theory for Efficiency Calculations From Annular Cascade Data," *Journal of Turbomachinery*, **119**(2), April, pp. 247–255.
- [30] Medic, G., Sharma, O. P., Jongwook, J., Hardin, L. W., McCormick, D. C., Cousins, W. T., Lurie, E. A., Shabbir, A., Holley, B. M., and Slooten, P. R. V., 2017, High Efficiency Centrifugal Compressor for Rotorcraft Applications Tech. Rep. NASA/CR—2014-218114/REV1, NASA Technical Report, United Technologies Research Center, East Hartford, Connecticut, October.
- [31] Kulkarni, S., Beach, T. A., and Skoch, G. J., 2013, "Computational Study of the CC3 Impeller and Vaneless Diffuser Experiment," In 49th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.
- [32] Kulkarni, S., Beach, T. A., and Skoch, G. J., 2013, "Computational Study of the CC3 Impeller and Vaneless Diffuser Experiment," In 49th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.
- [33] Skoch, G. J., Prahst, P. S., Wernet, M. P., Wood, J. R., and Strazisar, A. J., 1997, "Laser Anemometer Measurements of the Flow Field in a 4:1 Pressure Ratio Centrifugal Impeller," In Proceedings of ASME Turbo Expo 1997: Power for Land, Sea, and Air, Vol. 1 of *Aircraft Engine; Marine; Turbomachinery; Microturbines and Small Turbomachinery*, p. V001T03A049.

[34] Everitt, J. N., and Spakovszky, Z. S., 2012, "An investigation of stall inception in centrifugal compressor vaned diffuser," *J. Turbomach.*, **135**(1), 11, p. 011025.

APPENDIX: BLOCKAGE-MIXING-MODEL DERIVATION

A control volume analysis is carried out, assuming incompressible, fully reverse flow, no deviation at the LE ($\beta_2 = \kappa_2$), and isentropic jet flow in the passage, i.e. non-separated portion of the flow (denoted with the subscript j).

Aerodynamic blockage

Applying mass, momentum, and energy conservation in the control volume coinciding with the blade passage and using subscripts 1 for inflow and 2 for outflow properties, the blockage model is derived:

Mass flow conservation:

$$\rho W_1 \cos \beta_1 A_1 = \rho W_2 \cos \kappa_2 A_2 \overline{b_{aero}} \quad (17)$$

Conservation of rotary stagnation pressure in jet flow:

$$P_2 + \frac{1}{2} \rho (W_2^2 - U_2^2) = P_1 + \frac{1}{2} \rho (W_1^2 - U_1^2) \quad (18)$$

Streamwise momentum conservation along the full duct:

$$P_2 A_2 \cos \kappa_2 - P_1 A_1 \cos \kappa_1 + \left(\rho W_2^2 A_2 \overline{b_{aero}} \cos(\kappa_2) \right) - \left(\rho W_1^2 \cos(\beta_1 - \kappa_1) \cos \beta_1 A_1 \right) = \iiint \rho f_{cyl} \hat{l} dV \quad (19)$$

where, f_{cyl} is given by Eq. (9). Combining Eqns. (17) - (19) to eliminate P_2 and W_2 , Eq. (7) is derived.

Radial mixing - flow angle

For the control volume analysis the following assumptions are made alongside the aforementioned ones: mixing occurs outside the blade passage ($\Omega=0$), areas are only function of radius, i.e. strictly radial ($\varphi = 1$), free vortex flow, i.e. $V_\theta \sim 1/r$. Denoting with subscripts 2 and 3 the properties at the blade LE and the

downstream boundary of the control volume respectively (see Fig. 2), continuity and circumferential momentum are combined to obtain the mixed-out flow angle:

Continuity:

$$\rho V_3 A_3 \cos \alpha_{3,mix} = \rho V_2 A_2 \overline{b_{aero}} \cos \kappa_2 \quad (20)$$

Circumferential momentum:

$$\rho \sin \alpha_{3,mix} \cos \alpha_{3,mix} V_3^2 A_3 = \rho \sin \kappa_2 \cos \kappa_2 V_2^2 A_2 \overline{b_{aero}} - \iiint \rho \frac{V^2 \cos \alpha \sin \alpha}{r} dV \quad (21)$$

The integral term can be simplified by approximating the velocity in the middle of the volume using continuity and expressing the total volume as per Eqns. (22), (23). Note, that A_2, A_3 correspond to the entire annulus area.

$$\rho V_2 A_2 \overline{b_{aero}} \cos \kappa_2 = \rho V_{23} \frac{(A_2 + A_3)}{2} \cos \kappa_2 \quad (22)$$

$$dV \approx \frac{2\pi (r_2 + r_3)}{2} (r_2 - r_3) \frac{A_2}{2\pi r_2} = \frac{(r_2^2 - r_3^2)}{r_2} \frac{A_2}{2} \quad (23)$$

Combining the above equations and taking $\alpha = \kappa_2$ for simplicity (integral part contribution is small), Eq. (13) is derived.

Radial mixing - loss coefficient

The mixing loss is retrieved from the radial momentum equation:

Radial momentum:

$$\rho \cos \alpha_{3,mix}^2 V_3^2 A_3 + P_3 A_3 = \rho \cos \kappa_2^2 V_2^2 A_2 \overline{b_{aero}} + P_2 A_2 + \iiint \rho \frac{1}{r} \sin \alpha^2 V^2 dV \quad (24)$$

Solving for P_3 , substituting the integral as shown earlier and using Eq. (20) to replace V_3 , the loss coefficient is obtained as function of LE properties and the mixed-out flow angle $\alpha_{3,mix}$ as per Eq. (14).



DSSPACE

<https://dspace.org/>

A new body force-based modelling approach for centrifugal compressor post-stall aerodynamics

Lamprakis, Dimitrios; Righi, Mauro; Pachidis, Vassilios

2025-06-16

American Society of Mechanical Engineers (ASME)

<https://doi.org/10.1115/gt2025-153790>

Lamprakis D, Righi M, Pachidis V. (2025) A new body force-based modelling approach for centrifugal compressor post-stall aerodynamics. In: Proceeding of the ASME Turbo Expo 2025: Turbomachinery Technical Conference and Exposition, 16-20 Jun 2025, Memphis, Tennessee, USA, Volume 12, Volume 12: Turbomachinery: Multidisciplinary Design Approaches, Optimization, and Uncertainty Quantification; Turbomachinery: Radial Turbomachinery Aerodynamics; Turbomachinery: Unsteady Flows in Turbomachinery; Wind Energy; Special Lecture, Article number GT2025-153790