

Influence of Deterministic Stresses on Flow Prediction of a Low Speed Axial Flow Compressor Rear Stage

Y G Li and A. Tournlidakis

Department Power Engineering and Propulsion, School of Engineering, Cranfield
University, Bedfordshire MK43 0AL, UK

Abstract

Deterministic stresses are used to describe the unsteady effects on flow field in multistage axial flow compressors. In this paper, a circumferentially non-uniform mixing plane method implemented with deterministic stresses is used for the flow prediction inside the rear stage of an axial flow compressor in a multistage environment. The deterministic stress field is calculated with an overlapped solution approach for the same flow passage and implemented to the Navier-Stokes equations in order to provide continuous interfaces. The aerodynamic boundary condition is simplified with the application of a repeating stage model where only the mass flow rate and stage exit average static pressure are required.

Two computational cases for the third stage of the Cranfield Low Speed Research Compressor (LSRC), one with the mixing plane model and the second with the mixing plane model in combination with deterministic stresses, are carried out and compared with each other, and also with corresponding experimental data. It is shown that the implementation of the deterministic stresses is beneficial to the flow prediction but the

improvement in accuracy for low speed axial flow compressor rear stages is not significant.

Keywords: deterministic stress, axial flow compressor, mixing plane

NOTATION

A	area (m^2)
J	Jacobian of the Transformation
N	number of blades in a row
P	pressure (Pa)
U_m	midspan blade speed (m/s)
V	velocity (m/s)
$V^j, j = 1, 2, 3$	contravariant velocity components (m/s)
bf, f	body force (N)
blk	blockage
h	blade height (m)
k	turbulence kinetic energy
$\dot{m}$	mass flow rate (kg/s)
$\bar{r}$	position vector
r, θ, z	cylindrical coordinates
ρ, ρ	density (kg/m^3)
t	blade tip clearance height (m)

$u_i, i = x, y, z$	Cartesian velocity components (m/s)
$u_i, i = r, \theta, z$	cylindrical velocity components (m/s)
v_a, v_c	axial and tangential velocity components (m/s)
u, v, w	Cartesian velocity components (m/s)
$\bar{\Omega}$	rotational speed vector
ε	turbulent energy dissipation
$\xi^j, j = 1, 2, 3$	body-fitted computational coordinates ($\xi^1 = \xi, \xi^2 = \eta, \xi^3 = \zeta$)
λ_R	blockage weighting function
τ	stress tensor

Superscripts

'	fluctuation;
	new value after blocking correction

Subscripts

a	axial
bf	body force
c, θ	circumferential
dns	downstream of an interface
r	radial
ref	reference
s	static
t	stagnation
ups	upstream of an interface

x, y, z

in x, y, z direction

1 INTRODUCTION

There are three major steady state numerical methods for the flow prediction of multistage axial flow compressors, Chima et al. [4] and Adamczyk [2]. The first one is the successive analysis of blade rows where an analysis code for an isolated blade row is required. This method used by several researchers is numerically simple but it is difficult to accurately define the boundary conditions and keep the spanwise consistency of parameters at interfaces between blade rows. In addition, it ignores a lot of physical processes that may be important in real machines. The second one is the mixing plane model proposed by Denton [6] and Dawes [5], where spanwise consistency for major aerodynamic parameters are maintained at blade row interfaces although some physics may be missed. This type of model provides rapid solutions and satisfactory results for multistage axial flow compressors with large axial spacing working near peak efficiency. Unfortunately, large prediction errors may appear when the compressors operate at off-design conditions or the axial spacing is small. Adamczyk [1] introduced an average passage equation system for multistage turbomachinery flows, taking into account the unsteady effect into the steady state environment. Kirtley et al. [9] introduced a more general implementation of the average passage closure model and permitted the use of non-pure H-meshes.

Fully unsteady simulation of three-dimensional flows in compressors has been developed significantly in recent years and can offer great advantages over steady state methods. Unfortunately, applications of unsteady methods to multistage compressors are still far beyond the capability of today's computers. With Adamczyk's approach the unsteady effect is taken into account in the steady state modelling by time- and spatial-

averaging the governing equations. Such an approach for multistage simulations is within the capability of today's computers and although is more complicated than the mixing plane approach, is still much more economical numerically than the fully unsteady approach. The benefits of this approach were well proven in Rhie et al.'s work [17] where the mixing-plane/non-bodyforce approach (NASTAR-I) and the deterministic stress approach with body force (NASTAR-II) were applied to two three stage real world compressors which had high pressure ratios and small axial gaps. Significant prediction improvement in accuracy was obtained with NASTAR-II as compared to NASTAR-I.

In Adamczyk's average-passage equation system [1], one of the difficulties in solving the system is how to obtain the deterministic stresses to close the system. Adamczyk [1] was the first to suggest the idea of using overlapped solution domains, and Celestina et al. [3], Rhie et al. [17] and LeJambre et al. [11] implemented this idea in their calculations. Sondak et al. [18] presented a "lumped" deterministic stress (LDS) model where the deterministic stresses are modelled with an inviscid time dependent simulation, which was also used by other researchers, such as Hall [7]. With the overlapped domain approach the flow in an extended computational domain must be predicted with the influence of downstream blade row by implementing the body force and blade blockage effect of the downstream blade row. Rhie et al. [17] approximated the body force by projecting the blade surface pressure distribution to the blade chamber line but ignored the blade blockage.

At a working point near peak efficiency, the flow in the rear stages of multistage axial flow compressors always demonstrates a repeating nature in terms of the spanwise distribution of aerodynamic parameters. Based on this physical phenomenon, a repeating

stage model was first introduced by Li et al. [13] to simplify the boundary conditions for the flow prediction of rear stages of multistage axial flow compressors.

In this paper, a circumferentially non-uniform mixing plane model based on the idea of Denton [6] with the addition of deterministic stresses are used for the prediction of the third stage of the Cranfield Low Speed Research Compressor (LSRC). The flow field in the compressor is predicted with an overlapped solution approach based on Adamczyk's averaged-passage equation system [1] and Rhie et al.'s approach [17] while a different approach for the body force modelling was developed. The blade blockage effect on the flow field is approximated for the first time with mass continuity applied to each of the blocked computational cells. The repeating stage model due to Li et al. [13] is applied in the simulation in order to simplify the setting of the boundary conditions. The predicted results are compared with another prediction with the mixing plane model for the same compressor stage, with the same boundary conditions but without the implementation of deterministic stresses. The results are also compared with experimental data.

2 GOVERNING EQUATIONS AND NUMERICAL ALGORITHMS

It is assumed that the flow in a low speed axial flow compressor is three-dimensional, incompressible and steady in the relative frame of reference. Therefore, the governing equations are the continuity and momentum equations, i.e. the Navier-Stokes equations. In describing the flows in multistage turbomachinery, three averaging operators were applied to the Navier-Stokes equations [1]. The first operator, known as the ensemble averaging operator, decouples the unsteady flow features from the random fluctuations.

The second operator, the time-averaging operator, averages out unsteady time scales resulting from blade row aerodynamic interactions due to shaft rotation. The third operator is a passage-to-passage averaging operator which averages out the details of the passage-to-passage variation in the flow field. In the present study, the aperiodic components of flow unsteadiness are ignored so the third operator is not applied to the governing equations. Applying the ensemble- and time-averaging operators on the governing equations produces the following average-passage Navier-Stokes equations in non-orthogonal curvilinear coordinate system ξ^i , $i = 1, 2, 3$:

$$\frac{\partial}{\partial \xi^j} (J \bar{V}^j) = 0 \quad (1)$$

$$\frac{\partial}{\partial \xi^j} (J \bar{V}^j \bar{u}_i) = -\frac{\partial}{\partial \xi_j} \left(J \frac{\partial \xi_j}{\partial x_i} \bar{P} \right) + \frac{\partial}{\partial \xi_j} \left[J \frac{\partial \xi_j}{\partial x_k} (\bar{\tau}_{ij} - R_{ij}) \right] + J f_i \quad (2)$$

When the governing equations are solved in the flow field where only the random unsteadiness is taken into account, the Reynolds stress $\overline{\rho u_i' u_j'}$ will appear in the term R_{ij} which may be represented through the application of standard two-equation turbulence models, such as the k- ϵ turbulence model, Launder and Spalding [10]. This is the case when the mixing-plane model is applied.

If the governing equations are applied to the flow field where both the random and time unsteadiness are taken into account, another deterministic stress term $\overline{\hat{\rho} \hat{u}_i \hat{u}_j}$ will appear in R_{ij} together with the Reynolds stresses.

In order to calculate and implement the deterministic stresses conveniently, two of the non-orthogonal coordinates are aligned with the circumferential and radial coordinates respectively of the cylindrical coordinate system. The near wall regions are

modelled through the use of the standard wall function approach. In the solution procedure, Stone's Strong Implicit Procedure (SIP) [19] is used to solve the momentum equations first, subsequently a pressure correction equation [14] is solved to satisfy the continuity equation and update the velocity field and finally the k and ε equations are solved in turn.

3 REPEATING STAGE MODEL

In the multistage compressor environment, each rear stage of a multistage axial flow compressor behaves like a repeating stage of its neighbouring stages in terms of flow pattern. This assumption results in a repeating stage model [13] that can be used in flow simulations. It has the advantages that the inlet velocity profile becomes the result of the simulation rather than boundary condition and only the average static pressure downstream of the stage together with total mass flow rate are required as global input.

Figure 1 shows how the repeating stage model works. For a typical rear stage of a multistage axial flow compressor there are three mixing planes used to pass tangentially area-averaged information between neighbouring blade rows. It is assumed that the spanwise velocity profile at the inlet is set to be the same to that at the stage exit. The total mass flow rate is used to scale the inlet velocity profile in order to maintain mass conservation during global iterations. Similarly, the inlet k and ε profiles are set to be equal to those at the stage exit as they tend to be unchanged when the turbulent flow is fully developed. The way of setting the static pressure at the exit of the stage concerned is to maintain the same shape of spanwise static pressure distribution at the inlet and exit of the stage and to set the average static pressure at the stage exit to a pre-defined value. The

above setting also results in the similarity of the radial total pressure distribution at the inlet and exit of the stage. Further details are described in Li [12] and Li et al. [13].

4 MIXING PLANE MODEL

A circumferentially non-uniform mixing plane model [12] for flow prediction of multi-blade rows was used. The circumferential variation of aerodynamic parameters on both sides of the mixing plane is obtained with the idea of Denton [6]. With this model, frames of reference are fixed on the rotating blades for rotors and stationary blades for stators and the flow is supposed to be steady in the relative frame of reference. The mixing planes are chosen to be located midway between two successive blade rows.

In order to transfer the information from one side of the mixing plane to the other the following procedure is used (Figure 2). Firstly, extrapolation is used to obtain the values of variables at the mixing plane from upstream or downstream points of the same grid block. Then area averaging is used to get fluxes for different variables at different radial positions on both sides of the mixing plane. Finally, the fluxes for each variable on both sides of the mixing plane are forced to be equal by scaling, according to the propagating direction of the variable and the ratio of fluxes on both sides of the plane.

In the current study, the circumferentially non-uniform mixing plane model is only applied to the static pressure that propagates upstream and the k and ε that propagate downstream. A circumferentially uniform mixing plane is used to pass the area-averaged spanwise velocity profile, which propagates downstream, from one side of the mixing plane to the other for simplicity, without taking into account the circumferential variation of the velocity field at the inlet of a flow passage.

With the mixing plane model, circumferential variation exists on both sides of the plane for P_s , k and ε , and the consistency of mass, static pressure, velocity, k and ε at mixing plane is guaranteed. Unfortunately, although the mass continuity is conserved automatically because of the area averaged method used, the continuity of momentum and total pressure are not guaranteed at the mixing planes. The influence of these discontinuities may not be serious for the flow prediction of compressor stages working near peak efficiency but could be significant at off-design conditions. With the introduction of deterministic stresses momentum and total pressure across the interfaces become continuous; this will be discussed further in the following section.

5 DETERMINISTIC STRESS CALCULATION

In this paper, the deterministic stresses are calculated with an overlapped solution approach based on the idea of Adamczyk [1].

5.1 Overlapped Solution Domain

The computational domains for a compressor rear stage in a multistage environment are illustrated in Figures 3 and 4. The grid for each blade row starts axially from its upstream mixing plane and extends to the downstream mixing plane of its following blade row. In the relative frame of reference of a blade row, the flow is simulated with the influence of the downstream blade row by replacing the blade row with a body force and a blade blockage effect. The deterministic stress field is calculated in the extended solution domain. In overlapped computational domains, all the grid lines in $r-z$ (hub-to-tip) surfaces are aligned with each other and therefore the implementation of deterministic

stresses and body force becomes straightforward. The blade wakes can also develop in the extended computational domain with periodic boundary without further mesh requirements.

5.2 Body force Simulation

The body force is caused by the interaction between the flow and the blade row. It is associated with the inviscid and viscous forces acting on the blades. The influence of the body force is not only restricted to inside the area between the blades but is also extended upstream and downstream. Adamczyk [1] and Rhie et al. [17] introduced different methods of approximating the body force. In this paper, an effort has been made to simulate both the inviscid and viscous body forces in a different way [12], which will now be described.

It is assumed that the body force contributes to the pressure change and part of the momentum change of the flow. Another contribution to the momentum change comes from the impact of blade blockage, which will be discussed in the next section. Based on this assumption, the body force acting on a control volume is the pressure difference around the volume surface plus centrifugal force. It is also assumed that the centrifugal force generated by rotating blades only exists inside the area between blades.

The circumferentially area-averaged cylindrical body force components for the rotor 3 and stator 3 domains are shown in Figure 5. The axial body force contributes to the pressure change and part of the momentum change in the axial direction. The circumferential body force contributes to the turning of the flow in circumferential

direction and the radial body force is responsible for the pressure and mass flow rate distribution in the spanwise direction.

This approximation has the advantage that the body force, including inviscid and viscous forces, can be easily simulated by calculating the circumferentially area-averaged pressure gradients and centrifugal force (for rotor blade rows) from the current flow passage and then implementing these in the calculation of the upstream blade computational domain.

In the simulation procedure, the local pressure gradients in the current flow passage are firstly calculated in the three directions of the cylindrical coordinate system. Then circumferentially area-averaged pressure gradients at certain radii r are obtained with the following expressions:

$$\begin{aligned}(f_r)_r &= \frac{\int \left(\frac{\partial \bar{P}}{\partial r} \right)_r dA}{\int dA} + \bar{\rho} \bar{\Omega} \times (\bar{\Omega} \times \bar{r}), \\(f_\theta)_r &= \frac{\int \left(\frac{\partial \bar{P}}{r \partial \theta} \right)_r dA}{\int dA} \lambda_R, \\(f_z)_r &= \frac{\int \left(\frac{\partial \bar{P}}{\partial z} \right)_r dA}{\int dA}\end{aligned}\tag{3}$$

where λ_R is the local blockage weighting function defined by Adamczyk et al. [1]:

$$\lambda_R = 1 + \frac{(\theta_1 - \theta_2)N}{2\pi}\tag{4}$$

where the meaning of θ_1 and θ_2 are shown in Figure 5. The purpose of λ_R is to average out the tangential pressure gradient on the whole tangential length, including the area

occupied by blades, because this pressure gradient is supposed to be implemented in the calculation to change the tangential momentum in the upstream blade computational domain where the space of the blades is occupied by fluid.

Subsequently, the obtained area averaged pressure gradients are projected locally to the x, y and z directions:

$$\begin{aligned}(f_x)_r &= (f_r)_r \cos \theta - (f_\theta)_r \sin \theta \\(f_y)_r &= (f_r)_r \sin \theta + (f_\theta)_r \cos \theta \\(f_z)_r &= (f_z)_r\end{aligned}\tag{5}$$

The circumferentially averaged body force generated by a blade row can only produce the average circumferential effect on the extended flow field of its upstream blade row and therefore detailed tangential variation of the body force is smeared and the wake stretching effect can not be reproduced.

5.3 Blade Blockage Effect

Blade blockage has an obvious influence on the flow momentum inside the blocked area. When the flow area is partially blocked, the flow must run faster in the blocked direction in order to pass the unchanged mass flow rate. A method of implementing the influence of λ_R into the flow predictions was described by Adamczyk [1].

In this study a simplified approximation of blade blockage effect is introduced [12]. A typical control volume in the downstream computational domain of the blade row of interest is shown in Figure 5. When blockage occurs on a control volume, its effect is just like that due to flow passing through a control volume with a smaller cross area (which is drawn in dotted lines) where the projected flow areas are reduced in the z - and

r - directions and remain the same in the θ -direction. Assuming that the mass flow rates in the z , r and θ directions remain the same when the flow is blocked, the new velocity components in cylindrical coordinates are calculated based on mass continuity in each direction:

$$\begin{aligned}\dot{m}_z &= \rho A_z u_z = \rho (A_z \lambda_R) u'_z \\ \dot{m}_r &= \rho A_r u_r = \rho (A_r \lambda_R) u'_r \\ \dot{m}_\theta &= \rho A_\theta u_\theta\end{aligned}\tag{6}$$

and

$$\begin{aligned}u'_z &= u_z / \lambda_R \\ u'_r &= u_r / \lambda_R \\ u'_\theta &= u_\theta\end{aligned}\tag{7}$$

The updated velocity components are then projected to the Cartesian directions and the corresponding velocity components are:

$$\begin{aligned}u'_x &= u'_r \cos \theta - u'_\theta \sin \theta \\ u'_y &= u'_r \sin \theta + u'_\theta \cos \theta \\ u'_z &= u'_z\end{aligned}\tag{8}$$

The blade blockage effect contributes to another part of the axial momentum change inside the blade passages. Without this correction, the predicted momentum and total pressure in the extended computational domain covering the blade row following the current row will not be properly determined.

This approximation of the blockage effect is applied only after converged flow fields are achieved. The volume geometries used during the iterations are actually the

same as those in the computational domains instead of the blocked ones so the velocity components u_z , u_r and u_θ during the computation are the combination of $(u'_z \lambda_R)$, $(u'_r \lambda_R)$ and u'_θ respectively if expressed in cylindrical coordinates.

5.4 Deterministic Stresses

In three dimensions, the local deterministic velocity fluctuation can be expressed in a cylindrical coordinate system (Rhie et al. [16]) as

$$\hat{u}_i = \bar{u}_i - \bar{\bar{u}}_i, \quad (i = r, \theta, z) \quad (9)$$

where $\bar{u}_i$ is the ensemble-averaged velocity and $\bar{\bar{u}}_i$ is the time-averaged velocity calculated by:

$$\bar{\bar{u}}_i = \frac{\int_{\theta_2}^{\theta_3} \bar{u}_i dA}{\int_{\theta_2}^{\theta_3} dA} \quad (10)$$

where $dA = r d\theta dr$, θ_2 and θ_3 are pitchwise angles for two neighbouring blades as shown in Figure 5.

In order to include the deterministic stresses into the momentum equations, the calculated $\hat{u}_i$, ($i = r, \theta$) may be projected to the x and y directions:

$$\hat{u}_x = \hat{u}_r \cos \theta - \hat{u}_\theta \sin \theta \quad (11)$$

$$\hat{u}_y = \hat{u}_r \sin \theta + \hat{u}_\theta \cos \theta \quad (12)$$

Consequently, the tangentially area-averaged deterministic stresses may be calculated as:

$$\overline{\hat{\rho} \hat{u}_i \hat{u}_j} = \frac{\int \hat{\rho} \hat{u}_i \hat{u}_j dA}{\int dA}, \quad (i, j = x, y, z) \quad (13)$$

The calculated deterministic stress field from the rotor 3 and the stator 3 computational domains are then implemented into the stator 3 and the rotor 3 domains in Case 2, respectively (Figures 3 and 4), by shifting and interpolation, if required, where it is assumed that neighbouring stages have repeating flow features.

Deterministic stresses contribute to the correction of the momentum and make the spanwise continuity of momentum possible at interfaces, which can be expressed in incompressible flows by the following equation:

$$\int_A (\rho \bar{u}_i \bar{u}_j) dA = \left(\rho \bar{\bar{u}}_i \bar{\bar{u}}_j + \rho \hat{\hat{u}}_i \hat{\hat{u}}_j \right) A \quad (14)$$

Correspondingly, the area averaged total pressure is continuous as well at interfaces because of the compensation of the deterministic stresses applied:

$$\bar{P}_{t,ups} = \bar{P}_{t,dns} \quad (15)$$

where

$$\bar{P}_{t,ups} = \frac{\int_A \left[P_s + \frac{\bar{\rho}}{2} (\bar{u}^2 + \bar{v}^2 + \bar{w}^2) \right] dA}{\int_A dA} \quad (16)$$

$$\bar{P}_{t,dns} = \bar{\bar{P}}_s + \frac{1}{2} \bar{\rho} (\bar{\bar{u}}^2 + \bar{\bar{v}}^2 + \bar{\bar{w}}^2) + \frac{1}{2} \bar{\rho} (\hat{\hat{u}}^2 + \hat{\hat{v}}^2 + \hat{\hat{w}}^2) \quad (17)$$

6 APPLICATION, COMPARISON AND ANALYSIS

The compressor used in the present study is the Cranfield University 4-stage Low Speed Research Compressor (LSRC). The mechanical details of the compressor are reported by Swoboda et al. [21]. Briefly, the machine consists of four repeating stages and the blades are of a modern controlled-diffusion type which are designed to be representative of a

high speed compressor. In the case concerned, the rotor blades with 2% blade height tip clearance and shrouded stator blades are selected. The main characteristics of the compressor are described in Table 1.

The major focus is on the third stage of the compressor, as the first two stages serve to condition the flow into a steady and repeating behaviour and the fourth stage provides a typical downstream condition of a multistage environment. The experimental data were obtained during the Advanced Civil Compressor Aerodynamics (AC3A) project [20].

Two computational cases were carried out for the third stage of the LSRC, one with the mixing plane model called Case 1 (Figure 1) and the other with the mixing plane model plus deterministic stresses called Case 2 (Figures 3 and 4). In Case 2, two computations are carried out repeatedly - the Main Computation which is the mixing plane approach implemented with deterministic stresses and the Sub-Computation where the overlapped solution approach is used to obtain the deterministic stress field. For the Sub-Computation in Case 2, both the inlet and the exit boundaries come from the Main Computation, Figures 3 and 4. Numerical tests proved that certain grid density is required in order to obtain mesh independent solution while denser grids significantly increase the computational time. In this study, a two-block grid for the rotor flow passage with a total of 118,875 nodes for the mixing plane approach and 182,275 nodes for the overlapped solution approach are used for the rotor passage to take into account both the major flow and the tip clearance flow precisely. A single block grid with 91,875 nodes for the mixing plane approach and 140,875 nodes for the overlapped solution approach are used for the stator flow passage with shrouded blades. The grid dimensions and node numbers are

described in Table 1. The grid layouts of the third stage for both approaches are shown in Figures 6 and 7.

A Neumann boundary condition is imposed for the velocity at the exit and for pressure at the inlet of the computational domains. On all solid surfaces, a no-slip condition is used for velocity and a zero gradient condition for pressure is implemented. Periodic conditions are applied on both sides of upstream and downstream blade extensions. The cylindrical interface between two blocks is treated by means of halo cells extended from one block into the interior of the adjacent one. Interpolation is used to exchange information between the two blocks. The repeating stage model is applied to both cases to simplify the boundary setting, Figures 1, 3 and 4. The deterministic stresses are calculated and implemented in the flow fields as shown in Figures 3 and 4.

Two types of convergence criteria must be satisfied for a converged solution. One refers to the numerical convergence criterion for the governing equations for each blade row, which is satisfied when the residuals of all the governing equations are sufficiently small. The other is the physical convergence criterion for the global iterations, which is satisfied when the physical flow quantities become consistent at the interfaces. Under-relaxation is used to enhance the convergence both locally and globally.

The computational time requirement for each inner iteration is approximately 5.0 seconds for the mixing plane approach and 7.5 seconds for the overlapped solution approach on a DEC ALPHA. In Case 2 the Sub-Computation is carried out at every 5 global iterations of the Main Computation. Typically, a maximum number of 5,000 inner iterations is used locally for each blade row to satisfy the numerical convergence criteria of the governing equations and about 15 to 20 global iterations are required for global

convergence. Compared with the original mixing plane approach (Case 1), the mixing plane model combined with deterministic stresses (Case 2) requires around 60% more CPU time for the calculation due to the extended computational domains and the sub-computation for the deterministic stress fields.

Performance comparisons of the two computational cases of the LSRC third stage with experimental data are carried out at the operating point of peak efficiency, at Rotor 3 inlet, the interface between Rotor 3 and Stator 3, and Stator 3 exit. The comparisons are performed in terms of total pressure, static pressure and velocity components, based on available experimental data from the AC3A project [20] where total pressure, static pressure and flow angle distributions were directly measured whereas the velocity components were calculated from them.

The predicted area-averaged static and total pressure distributions in the axial direction of the compressor stage are presented in Figure 8. Compared with the experimental data, good agreement between predictions and experimental data is shown in the figure although small discrepancies exist. Sudden changes of the pressure near the blade leading and trailing edges reflect the strong interaction between the flow and the blades. The total pressure difference on both sides of the mixing plane is very small, showing that the resulting discontinuity of the momentum from the mixing plane model in Case 1 is negligible at peak efficiency. It also shows that there is very little difference between the two predictions in most of the axial positions except in the upstream extension of the rotor blade row where Case 2 gives a better result.

The predicted spanwise distribution of velocity components from both computational cases match very well with the experimental data for most part of the

blade span at three axial stations (Figure 9) although with some discrepancies in the area between 80% and 100% span near the blade tip. These discrepancies are very likely due to numerical inaccuracies in resolving the complex, unsteady tip clearance flow, which is a common problem of most CFD methods. Very little difference in the spanwise velocity distributions between the two predictions is shown.

Both computational cases give good predictions for the spanwise static pressure distributions at the stage inlet, the interface between the two blade rows and the stage exit with a slight over-estimation of static pressure at the inlet of Rotor 3 and under-estimation of static pressure at its exit. These discrepancies also contribute to the discrepancies in the total pressure at the same area. At the Rotor 3 inlet, Case 2 provides a better prediction of the spanwise static pressure distribution than Case 1. The sudden drops of the experimental static pressure near the outer casing are not physical and are susceptible to measurement error due to the presence of the solid walls [8].

The spanwise total pressure distributions are well-predicted for both cases as compared with experimental data except for the area between 80% and 100% span, whereas Case 2 gives better prediction of the spanwise total pressure distribution in the area between 0% and 80% blade height. The compressor involved in the present study is a low speed research compressor, the flow is assumed to be incompressible and the total pressure is calculated using static pressure and velocity values so the discrepancy of total pressure near the blade tip is caused by the inaccurate prediction of velocity distribution in that area.

Compared with Case 1, an obvious improvement of the static and total pressure distribution and very slight improvement in the velocity distribution in the upstream

extension of Rotor 3 are obtained in Case 2 due to the implementation of the deterministic stresses. Almost no improvement is obtained for all the parameters downstream the leading edge of Rotor 3. Generally speaking, the benefits obtained from the implementation of deterministic stresses for the flow and performance prediction of the whole stage is not significant. This may be due to the magnitude of the deterministic stresses illustrated in Figures 11 and 12 which present the principle tangentially area-averaged deterministic stress components. The deterministic stresses obtain maximum values near the trailing edge of the upstream blade row, deteriorate rapidly downstream and become very small at the interfaces. This means that, for this low pressure compressor, the non-uniformity of the flow generated by an upstream blade row is mixed out rapidly and the benefit from the implementation of the deterministic stresses is not obvious, as can be seen from the performance comparison in Figures 8, 9 and 10.

7 CONCLUSIONS

The overlapped solution approach is an effective steady-state method to simulate deterministic stresses. The developed method for simulating the body force with pressure gradients is effective in the simulations and simple in programming. The blade blockage effect is approximated successfully in the compressor flow simulations for the first time with mass continuity applied to each of the blocked computational cells and good agreement of axial pressure and velocity distributions have been obtained in the overlapped computational domain.

The influence of deterministic stresses on the flow prediction is not significant for low speed axial flow compressors. The application of the repeating stage model

effectively simplifies the boundary conditions and provides satisfactory results. The predicted performance with both the mixing plane model and the mixing plane model in combination with deterministic stresses match very well with the experimental data.

ACKNOWLEDGEMENT

The authors wish to thank the Laboratory of Thermal Turbomachines of the National Technical University of Athens for their kind permission to use the “ELISA” code at the initial phase of this project.

REFERENCES

- 1 **Adamczyk, J. J.** Model Equation for Simulating Flows in Multistage Turbomachinery. NASA TM-86869; Also ASME paper 85-GT-226, 1984.
- 2 **Adamczyk, J. J.** Aerodynamic Analysis of Multistage Turbomachinery Flows in Support of Aerodynamic Design. ASME paper 99-GT-80, 1999.
- 3 **Celestina, M. L., Mulac, R. A. and Adamczyk, J. J.** A Numerical Simulation of the Inviscid Flow Through a Counterrotating Propeller. *Journal of Turbomachinery*, 1986, **108**, pp.187-193.
- 4 **Chima, R. V.** Calculation of Multistage Turbomachinery Using Steady Characteristic Boundary Conditions. AIAA paper 98-0968, 1998.
- 5 **Dawes, W. N.** Toward Improved Throughflow Capability: The Use of Three-Dimensional Viscous Flow Solver in a Multistage Environment. *Journal of Turbomachinery*, 1992, **114**, pp.8-17.
- 6 **Denton, J. D.** The Calculation of Three-Dimension Viscous Flow Through Multistage Turbomachineries. *Journal of Turbomachinery*, 1992, **114**, pp.18-26.
- 7 **Hall, E. J.** Aerodynamic Modelling of Multistage Compressor Flowfield - Part 2: Modelling Deterministic Stresses. ASME paper 97-GT-345, 1997.
- 8 **Howard, M. A., Ivey, P. C., Barton, J. P. and Young, K. F.** Endwall Effects at two Tip-Clearances in a Multistage Axial Flow Compressor with Controlled Diffusion Blading. *Journal of Turbomachinery*, 1994, **116**, No.4, pp.635-647.
- 9 **Kirtley, K. R., Turner, M. G. and Saeidi, S.** An Average Passage Closure Model for General Meshes. ASME paper 99-GT-77, 1999.

- 10 **Launder, B. E. and Spalding, D. B.** The Numerical Computation of Turbulent Flows. *Int. Com. Meth. in Appl. Meth. Engg.*, 1974, **3**, pp.269-289.
- 11 **LeJambre, C. R., Zacharias, R. M., Biederman, B. P., Gleixner, A. J. and Yetka, C. J.** Development and Application of a Multistage Navier-Stokes Flow Solver. Part II: Application to a High Pressure Compressor Design. ASME paper 95-GT-343, 1995.
- 12 **Li, Y. G.** Three-Dimensional Flow and Performance Simulation of Multistage Axial Flow Compressors. 2000, PhD Thesis, Cranfield University, England.
- 13 **Li, Y. G., Tournlidakis, A. and Elder, R.L.** Three-Dimensional Performance Prediction of Multistage Axial Flow Compressors with a Repeating Stage Model. ASME paper 99-GT-290, 1999.
- 14 **Patankar, S. V. and Spalding, D. B.** A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows. 1972, *International Journal of Heat and Mass Transfer*, **15**, pp.1787-1806.
- 15 **Politis, E. S., Giannakoglou, K. C., and Papailiou, K. D.** Axial Compressor Stage Analysis Through a Multi-Block 3-D Navier-Stokes Solution Method. ASME paper 97-GT-93, 1997.
- 16 **Rai, M. M.** Unsteady Three-Dimensional Navier-Stokes Simulation of Turbine Rotor-Stator Interaction. 1989, *Journal of Propulsion and Power*, **5**, No.3, pp.307-319.
- 17 **Rhie, C. M., Gleixner, A. J., Spear, D. A., Fischbery, C. J. and Zacharias, R. M.** Development and Application of a Multistage Navier-Stokes Solver. Part I:

Multistage Modelling Using Bodyforce and Deterministic Stress. ASME paper 95-GT-342, 1995.

- 18 **Sondak, D. L., Dorney, D. J. and Davis, R. L.** Modelling Turbomachinery Unsteadiness with Lumped Deterministic Stresses. AIAA paper 96-GT-2570, 1996.
- 19 **Stone, H. L.** Iterative Solution of Implicit Approximations of Multidimensional Partial Differential Equations. 1968, *SIAM J. Numerical Analysis*, **5**, pp.530-558.
- 20 **Swoboda, M., Ivey, P. C., Wenger, U. and Gümmer, V.** An Experimental Examination of Cantilevered and Shrouded Stator in a Multistage Axial Compressor. ASME paper 98-GT-282, 1998.
- 21 **Swoboda, M., Wenger U., and Gümmer, V.** BRITE EURAM (Project AC3A) Blading Design Report for the 4-Stage CU (Cranfield University) Low Speed Research Compressor (LSRC). 1996, Brite Euram Report 11-BRR-02.

Parameter	Rotor	Stator
Mass flow rate (kg/s)	12.04	12.04
Rotating speed (rpm)	1100	1100
Tip clearance (t/h)	2.0%	0.0%
Number of blades	75	96
Aspect ratio	1.36	1.36
Space/chord	0.71	0.55
Grids (for overlapped solution approach)	35x35x115 (main) 40x9x115 (tip)	35x35x115
Total node number	182,275	140,875
Grids (for mixing plane approach)	35x35x75 (main) 40x9x75 (tip)	35x35x75
Total node number	118,875	91,875

Table 1: Details of LSRC Rotors and Stators

List of captions:

- Figure 1: Repeating Stage Model (Case 1) [13]
- Figure 2: Mixing Plane Model [6]
- Figure 3: Case 2: Main Computation
- Figure 4: Case 2: Sub-Computation
- Figure 5: Body force and Blade Blockage Effect
- Figure 6: Grid Layout for Case 1 and the Main Computation of Case 2
- Figure 7: Grid Layout for the Sub-Computation of Case 2
- Figure 8: P_t and P_s in Axial Direction
- Figure 9: Comparison of Spanwise Velocity Distribution at Rotor 3 Inlet, Rotor 3 Exit and Stator 3 Exit
- Figure 10: Comparison of Spanwise Pressure Distributions at Rotor 3 Inlet, Rotor 3 Exit and Stator 3 Exit
- Figure 11: Circumferentially Area-Averaged Deterministic Stress Components at Rotor 3 Downstream
- Figure 12: Circumferentially Area-Averaged Deterministic Stress Components at Stator 3 Downstream

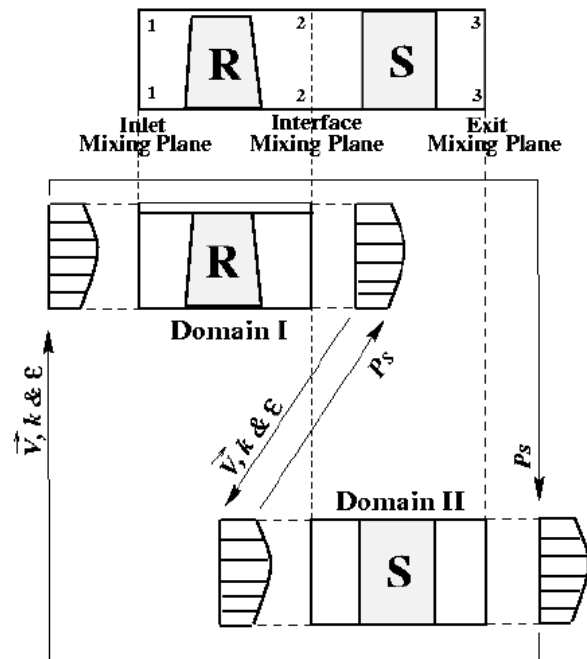


Figure 1: Repeating Stage Model (Case 1) [13]

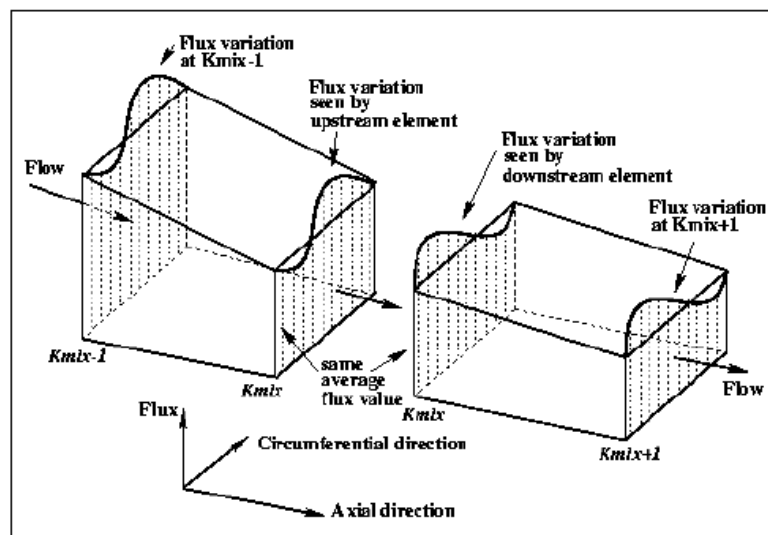


Figure 2: Mixing Plane Model [6]

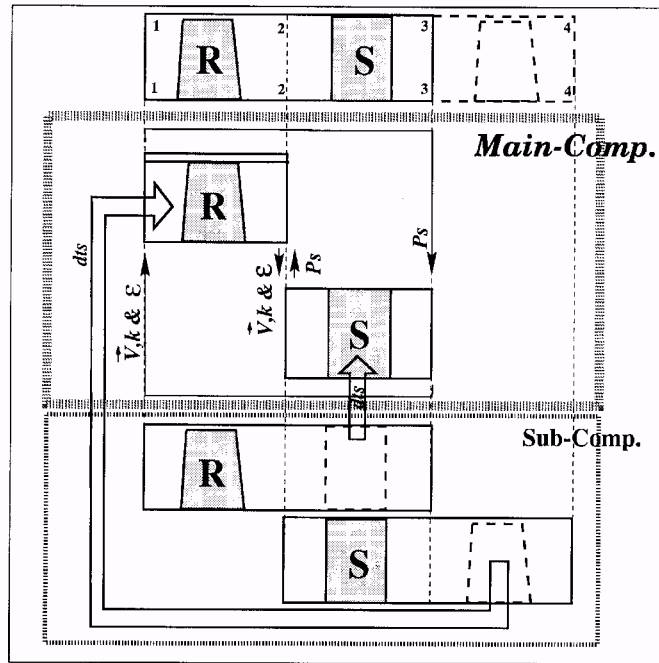


Figure 3. Case 2: Main computation.

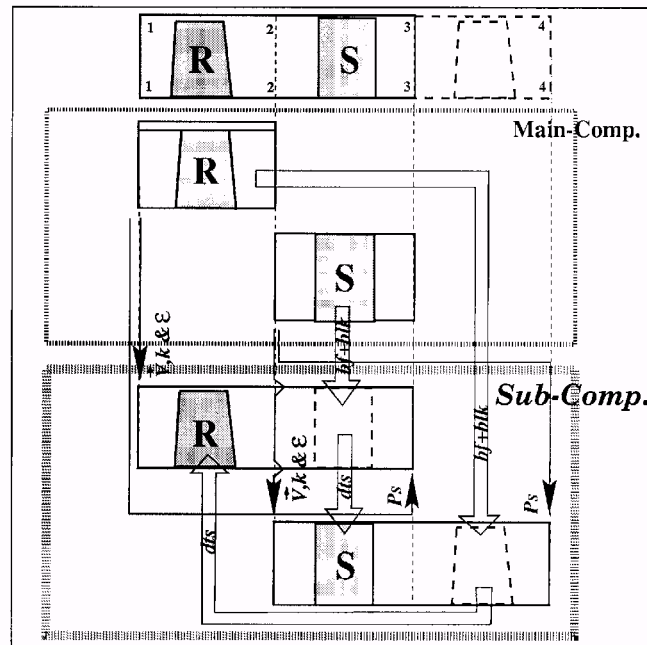


Figure 4. Case 2: Sub-computation.

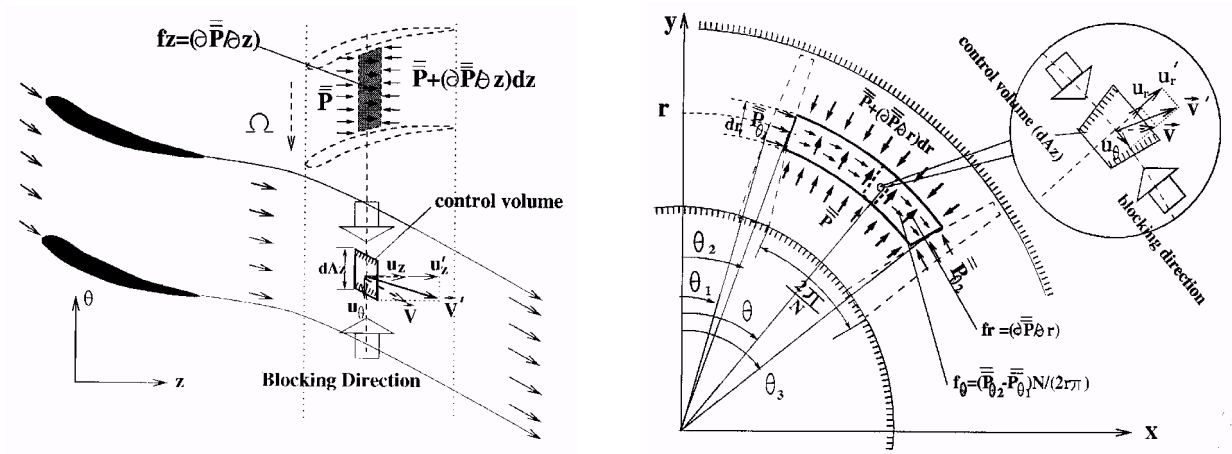


Figure 5. Body force and blade blockage effect.

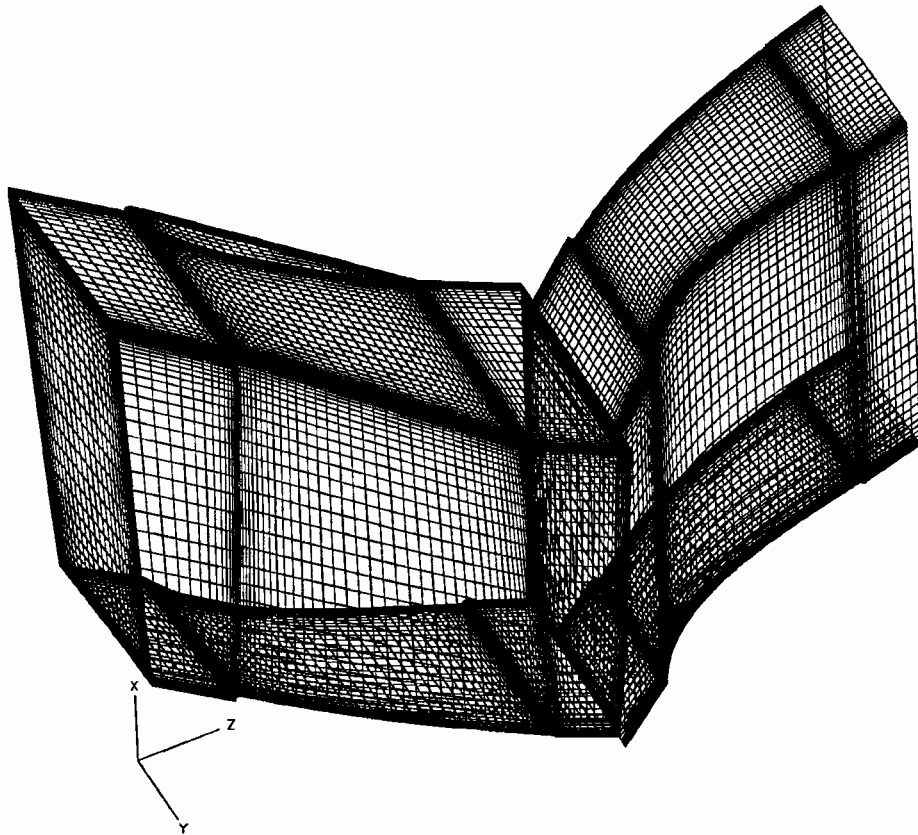


Figure 6. Grid layout for Case 1 and the main computation of Case 2.

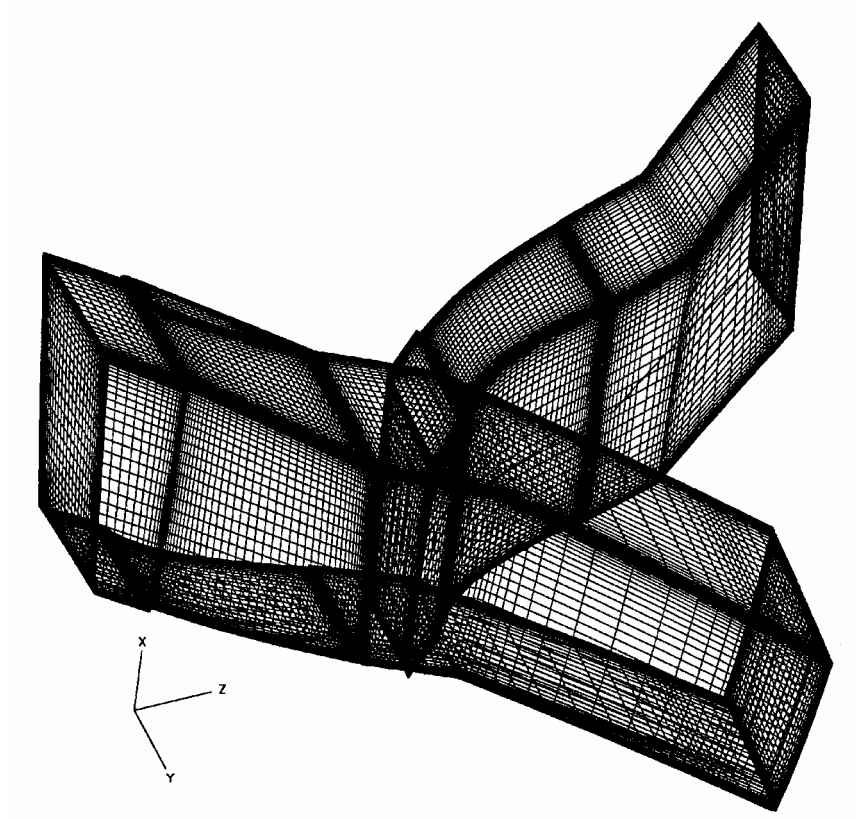


Figure 7. Grid layout for the sub-computation of Case 2.

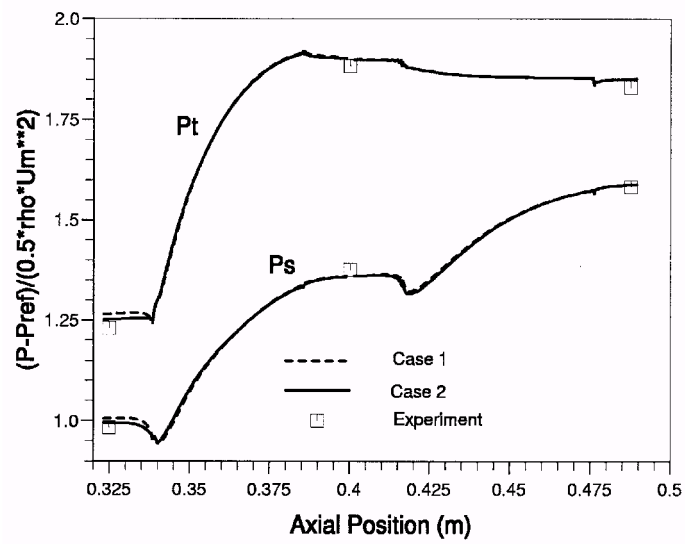


Figure 8. P_t & P_s in axial direction.

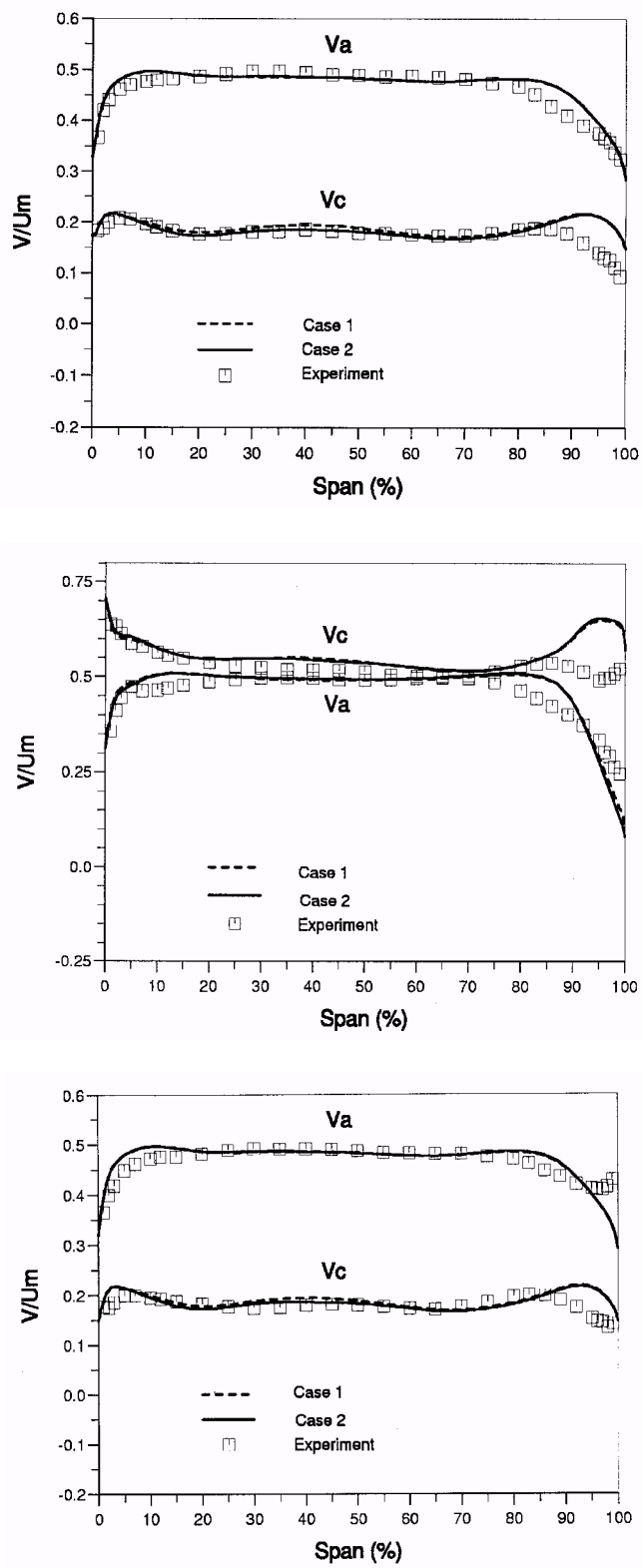


Figure 9. Comparison of spanwise velocity distribution at Rotor 3 inlet, Rotor 3 exit and stator 3 exit.

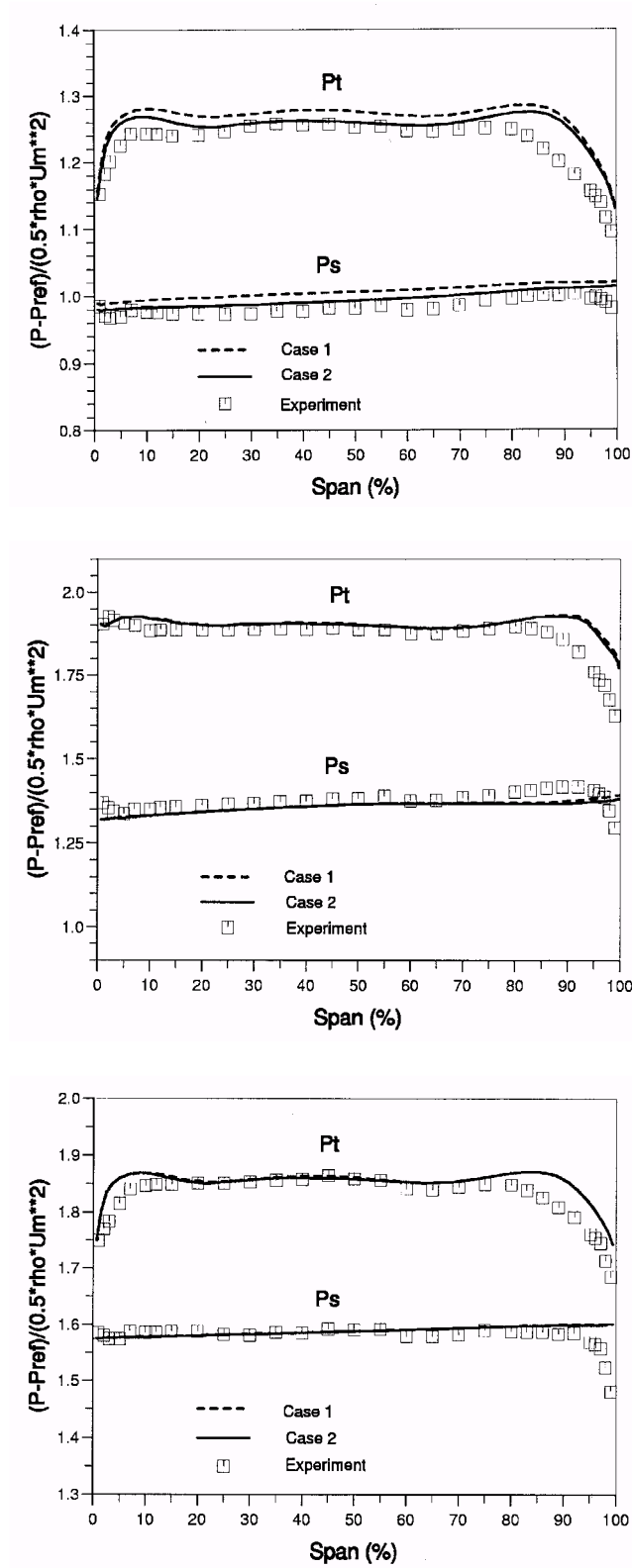


Figure 10. Comparison of spanwise pressure distribution at Rotor 3 inlet, Rotor 3 exit and Stator 3 exit.

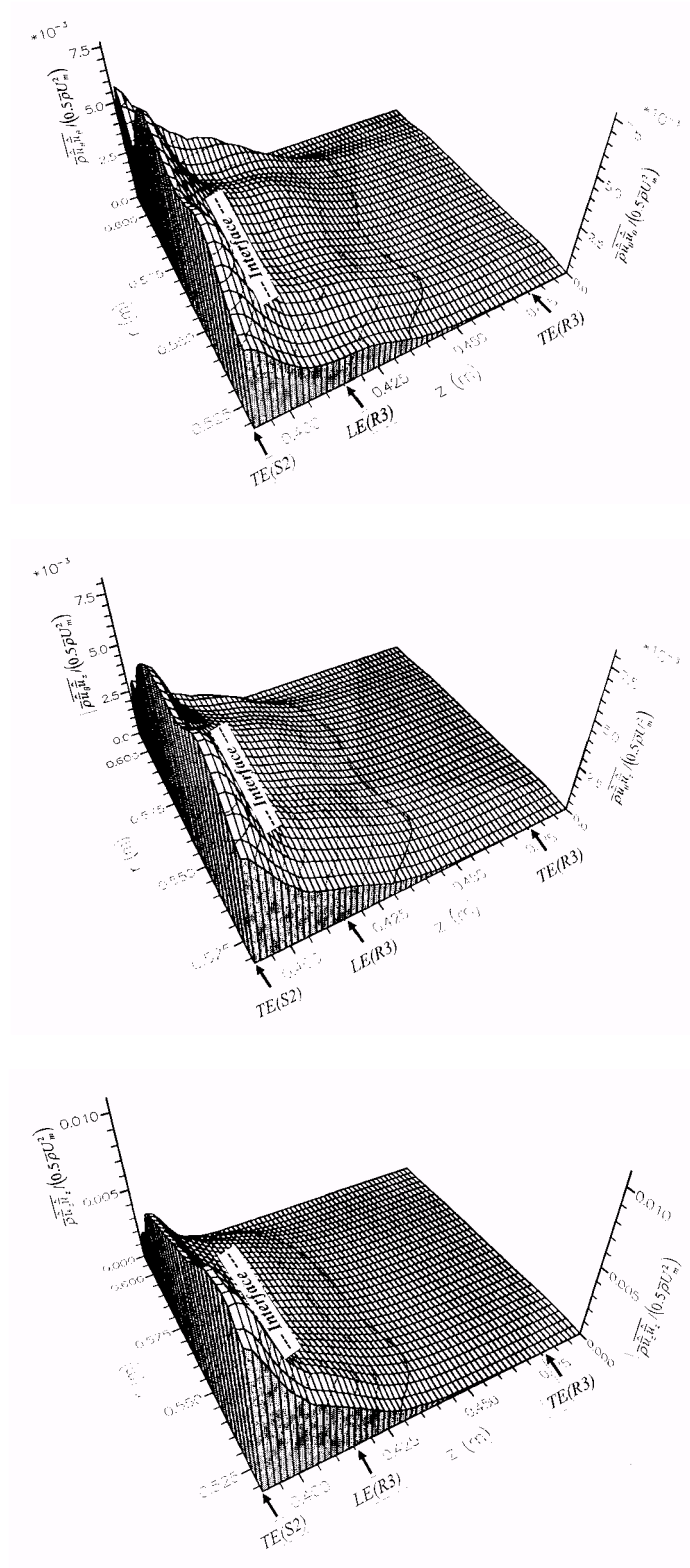


Figure 11. Circumferentially area-averaged deterministic stress components at Rotor 3 downstream

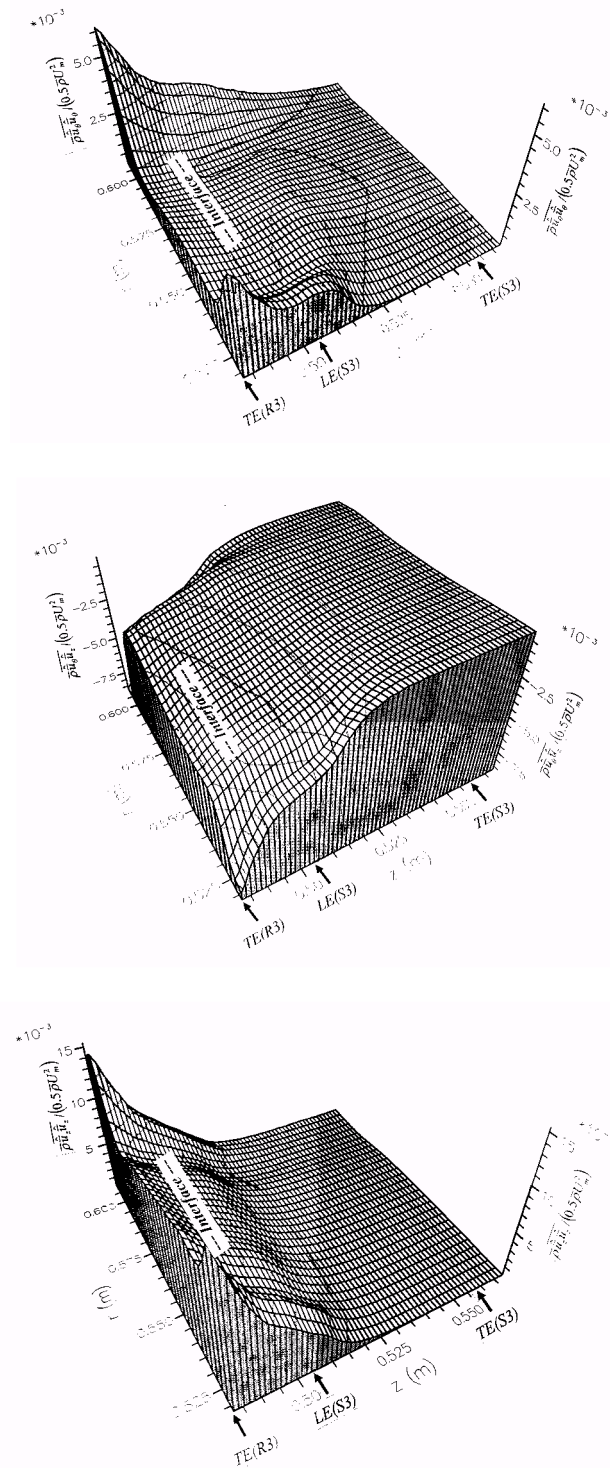


Figure 12. Circumferentially area-averaged deterministic stress components at Stator 3 downstream



DSpace

<https://dspace.org/>

Influence of deterministic stresses on flow prediction of a low-speed axial flow compressor rear stage.

Li, Y. G.; Tournlidakis, A.

2001-10-02T00:00:00Z

Professional Engineering Publishing

<http://dx.doi.org/10.1243/0957650011538811>

Y G Li, A Tournlidakis. Influence of deterministic stresses on flow prediction of a low-speed axial flow compressor rear stage. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, Volume 215, Number 5 (October 2, 2001), pp. 571-583,