A QUASI-ONE DIMENSIONAL MODEL FOR AXIAL COMPRESSORS

Olivier Adam*, Olivier Léonard†
Turbomachines et Propulsion
Université de Liège
4000 Liège, Belgique
o.adam@ulg.ac.be

ABSTRACT

The object of the present paper is to present a fast and reliable CFD tool that is able to simulate stationary and transient operations of multistage axial compressors. The computational domain is the compressor flow path, using a row-by-row, quasi-one-dimensional representation of the machine at mid-span.

This analysis tool is based on an adapted version of the Euler equations solved by a time-marching, finite-volume method. The basic Euler equations have been extended by including source terms expressing the blade-flow interactions. The source terms are determined using the velocity triangles for each blade row, at mid-span. The losses and deviations undergone by the fluid in each blade row are supplied by correlations.

The resulting flow solver is a performance prediction tool based only on compressor geometry. It offers the possibility of exploring the entire characteristic map of a compressor before its construction. Its efficiency in terms of CPU time makes it possible to use it as a fast design tool by coupling it to an optimization algorithm.

In this paper, this CFD tool (called Quads hereafter) has been applied to two test cases. Calculated characteristic curves are presented and compared to experimental ones. A correlation tuning process is described for an undocumented family of blade profiles.

NOMENCLATURE

Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>e</td>
<td>specific internal energy</td>
</tr>
<tr>
<td>F</td>
<td>conservative flux vector</td>
</tr>
<tr>
<td>$F_b$</td>
<td>blade force</td>
</tr>
<tr>
<td>$h$</td>
<td>specific enthalpy</td>
</tr>
<tr>
<td>$hv$</td>
<td>path flow local height</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$i$</td>
<td>incidence</td>
</tr>
<tr>
<td>$m$</td>
<td>curvilinear streamline coordinate</td>
</tr>
<tr>
<td>$p$</td>
<td>static pressure</td>
</tr>
<tr>
<td>$Q$</td>
<td>source term vector</td>
</tr>
<tr>
<td>$q_m$</td>
<td>mass flow</td>
</tr>
<tr>
<td>$r$</td>
<td>radius</td>
</tr>
<tr>
<td>$S$</td>
<td>compressor cross-sectional area</td>
</tr>
<tr>
<td>$SW$</td>
<td>shaft work</td>
</tr>
<tr>
<td>$t$</td>
<td>time</td>
</tr>
<tr>
<td>$TC$</td>
<td>blade tip clearance</td>
</tr>
<tr>
<td>$U$</td>
<td>conservative variable vector</td>
</tr>
<tr>
<td>$U_r$</td>
<td>tangential rotation speed</td>
</tr>
<tr>
<td>$V$</td>
<td>absolute velocity</td>
</tr>
<tr>
<td>$Vol$</td>
<td>blade row volume</td>
</tr>
<tr>
<td>$W$</td>
<td>relative velocity</td>
</tr>
<tr>
<td>$x$</td>
<td>abscissa</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$</td>
<td>absolute flow angle</td>
</tr>
<tr>
<td>$\beta$</td>
<td>relative flow angle</td>
</tr>
<tr>
<td>$\delta$</td>
<td>deviation</td>
</tr>
<tr>
<td>$\delta^*$</td>
<td>boundary layer displacement thickness</td>
</tr>
<tr>
<td>$\phi$</td>
<td>blade camber angle</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>cascade solidity</td>
</tr>
<tr>
<td>$\Omega$</td>
<td>rotation speed</td>
</tr>
<tr>
<td>$\varpi$</td>
<td>loss coefficient</td>
</tr>
<tr>
<td>$\Delta\beta$</td>
<td>fluid deflexion</td>
</tr>
</tbody>
</table>

Subscripts

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$x$</td>
<td>axial</td>
</tr>
<tr>
<td>$r$</td>
<td>radial</td>
</tr>
<tr>
<td>$\theta$</td>
<td>tangential</td>
</tr>
<tr>
<td>1</td>
<td>blade leading edge</td>
</tr>
<tr>
<td>2</td>
<td>blade trailing edge</td>
</tr>
<tr>
<td>$\infty$</td>
<td>blade row reference direction</td>
</tr>
<tr>
<td>$in$</td>
<td>compressor inlet</td>
</tr>
<tr>
<td>$out$</td>
<td>compressor outlet</td>
</tr>
</tbody>
</table>

Superscripts

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\circ$</td>
<td>total condition</td>
</tr>
<tr>
<td>$*$</td>
<td>minimum-loss condition</td>
</tr>
</tbody>
</table>
INTRODUCTION

The performances of a compressor must be studied and analyzed before it is tested on the bench or in a gas turbine engine. This study provides the designer with guidelines for the choice of the many design parameters and for optimizing the final configuration of the machine. It also allows to test (in a virtual way) the correct operation of the machine during critical manoeuvres. This exploration phase is based on the numerical modelling of the operation of the compressor.

This simulation tool must not only be accurate, but also fast and robust, whether it is used for design, analysis or validation purposes.

Thanks to the progress of the simulation methods and computing power, it is now possible to develop and to use compressor models based on mass, momentum and energy balances applied to a one-dimensional discretization of the compressor, blade row per blade row. This approach makes it possible to describe with a high degree of accuracy the exchanges of mass, energy and momentum within the machine, while reducing the need for information of an empirical nature such as characteristic curves, which are often not available, or which are generic and not very accurate. Moreover many physical phenomena such as bleedings, heat transfers, water ingestion, fouling... can be easily integrated into this kind of simulation due its very general formulation.

In this context, the present paper is based on the application of current CFD techniques to simulate the performance of an axial compressor based only on the knowledge of its geometry and functional parameters. The flow path is discretized in the streamwise direction only, with several elementary cells in each blade row or between blade rows. The equations of mass, momentum and energy are discretized on the resulting mesh and integrated with state-of-the-art CFD techniques. Various source terms are added to account for the physical phenomena associated with an axial compression system. Losses and deviations are obtained from empirical correlations. This approach confers the method with a wide range of application, in the predictive or analysis fields.

Two experimental compressor test-cases have been submitted to this computer code. The first one is a low-speed one-stage axial compressor based on DCA and circular-arc C4 profiles, which are well documented. The second test-case is a three-stage axial compressor with non-standard profiles, and the correlations for the losses and the deviations must be tuned to obtain a good agreement with the experimental data.

In the following sections, the choice of the model and the development of the set of equations are presented.

MODEL SELECTION

This project arose from an industrial need for a stationary, transient and dynamic simulation of axial compressor operation. It was postulated that the characteristic maps were \textit{a priori} unknown. Therefore, to simulate its performance, the method had to be based on compressor geometry alone.

The tool had to be fast, because of the possible unsteady aspect and the intensive use needed to draw a complete characteristic map. Throughflow 2D and full 3D simulations were then let apart.

The streamline curvature algorithms were also avoided, because of their restricted field of application. Indeed, one of the goals of the project was to yield a simulation of stalled or reversed flows.

On the other hand, over-simplified models do not offer the versatility of finite-volume solvers. Analytical [1][2] and lumped-volume (“0D”) models may simulate a wide range of operating conditions but their extension to technological effects is difficult.

Most of the analytical, 0D or 1D models aimed towards the detailed description of compressor flows make use of single-stage characteristic maps. The steady-state maps must be user-supplied [3] or estimated, which limits the application of these models as predictive tools [4]. Moreover, stage characteristics are established through expensive rig tests and cover only a part of the compressor operational domain.

The prediction problem has been addressed by several authors. Davis proposed a generation of individual stage maps from the compressor geometry [5]. The pseudo-experimental data are then used in combination with a classical 1D analysis code. This technique lacks flexibility and is also an additional, time-consuming layer in the simulation. Some methods seem more attractive, as they make use of the compressor geometry to estimate the velocity triangles at mid-span, on each blade row. Empirical information is introduced into the model through correlations describing the losses and deviations associated with the pressure rise inside the blade rows [6][7].

As a result, an original model was defined. It has been chosen to model the mid-span flow, which is supposed to be axisymmetric. The radial compo-
The development of the model equations, written in cylindrical coordinates, is based on the application of conservation principles to an elemental control volume.

The set of equations resulting from the above assumptions is similar to the \((x, \theta)\) model of Reddy and Nayani \[8\], with the flow being viewed as axisymmetric. The model contains two momentum equations: besides the axial momentum conservation, a second equation accounts for angular momentum variations.

In vector, non-dimensional, conservative form, these equations are written as

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}}{\partial x} = \mathbf{Q}$$  \hspace{1cm} (1)

$$\mathbf{U} = \begin{bmatrix} \rho S \\ \rho V_x S \\ \rho V_y S \\ \rho e^S \end{bmatrix} \quad \mathbf{F} = \begin{bmatrix} \rho V_x S \\ (p + \rho V_x^2)S \\ \rho V_x V_y S \\ \rho h^S \end{bmatrix}$$  \hspace{1cm} (2)

\(\mathbf{Q}\) is the vector of source terms. These source terms introduce in the Euler equations the effects of area and mean radius variations, losses and deviations undergone by the fluid through the blade rows. The vector has been divided into three distinct contributions, categorized according to their physical meaning:

$$\mathbf{Q} = \mathbf{Q}_b + \mathbf{Q}_f + \mathbf{Q}_g$$  \hspace{1cm} (3)

These source terms mainly depend on the velocity triangles and the method to evaluate these is described hereafter. The source term components are then detailed.

**Method of the velocity triangles**

The solver is based on a time-marching algorithm and at each time step, the velocity triangles are drawn at the mid-radius on each blade row. The predictive capability of the code derives fundamentally from this process.

Starting from the blade geometry at mid-span, the conservative variables of the current solution are combined with empirical information brought by correlations to fully determine the velocity triangle.

The blade material angles are known at the leading and trailing edges, as are the fluid angles and velocity on the blade leading edge.

![Figure 1: Velocity triangles over a blade](image)

The accuracy of the simulation is tightly related to the adequacy of the correlations with the compressor hardware. The primary losses and deviation correlations, particularly, must match the blade profile types. Each manufacturer has its own, confidential, correlations, and it is often difficult to render the operation of a modern compressor by means of the correlations available in the open literature. Tuning parameters have therefore been integrated into standard correlations.

The NASA method is used to evaluate the reference minimum-loss incidence, and the reference deviation is given by Carter’s rule \[9\]:

$$\gamma^* = K_{sh} \cdot K_{ij} \cdot (\theta_o) + n \cdot \phi$$  \hspace{1cm} (4)

$$\delta^* = \frac{m \phi}{\sqrt{\sigma}}$$  \hspace{1cm} (5)

The shape, thickness and camber coefficients, as well as the zero-camber incidence angle, have been fitted from the NASA data.
The “standard” off-design deviation correlation is the one from Creveling [10]. It has been chosen for its good performance on a wide range of profiles [11][12].

\[
\frac{\delta - \delta^*}{\Delta \beta^*} = f \left( \frac{i - i^*}{\Delta \beta^*} \right) \tag{6}
\]

The function \( f \) has been fitted from experimental data by two quadratic functions.

The blade row flow angles and mass flow are known from the previous evaluations. The fluid axial speed at the blade row exit is iterated until these conditions are matched, with the primary losses being evaluated for each exit speed.

The reference loss coefficient is evaluated through the classical correlation of Koch and Smith [13]. This very complete correlation includes a large number of physical parameters and has proven to produce accurate results.

The off-design profile loss coefficient, when using the standard open-literature correlations, is deduced from the results of Creveling [10].

\[
\sigma - \sigma^* = c_{m(M)} \cdot (i - i^*)^2 \tag{7}
\]

In reason of its particular shape, the coefficient \( c_m \) curve of Creveling has been fitted by a Chebyshev polynomial approximation. This same curve can be tuned through optimization parameters if needed.

An additional loss coefficient contribution renders the complex loss mechanisms associated with the secondary flows. It is based on a simple relationship, suggested by Howell [14] and still widely used [15][16].

\[
\sigma_x = 0.018 \cdot C_L \cdot \sigma \cdot \cos^2 \frac{\beta_1}{\cos^3 \beta_m} \tag{8}
\]

The lift coefficient is estimated through the flow angles of the velocity triangles.

Although the system of equations (1) describes a mean flow for the whole compressor channel height, spanwise variations of losses and deviations must be included in the simulation to achieve a minimum of trustworthiness.

Wall effects for rotors and stators, such as flow overturning or tip clearance losses [15], have been introduced into the correlations in the form of loss coefficient and deviation corrections. According to Roberts et al [17][18], parametric spanwise curves have been averaged along the blade height to obtain mean contributions, which depend on operational parameters like end-wall boundary layer thicknesses, blade camber and solidity, channel aspect ratio, or wheel tip clearance.

A sample spanwise curve, as described by Roberts et al, is presented in figure 2 for a rotor. This graph represents the variations of the local deviation angle from the “2D” angle, predicted by the correlations based on the blade element theory.

![Figure 2: Spanwise variation of rotor deviation](image)

Assuming a “standard” axial speed distribution represented in figure 3, the loss and deviation curves proposed by Roberts et al, for rotors and stators, have been integrated using the axial speed as a weighting factor.

![Figure 3: Normalized axial speed distribution](image)

The maximum level of axial speed has been set up according to the axial speed variable, which is representative of the average flow through the compressor:

\[
\int_{r_{hub}}^{r_{tip}} V_x(r) \cdot dr = \frac{7 \cdot h v}{8} V_{max} = \bar{V}_x \cdot h v \tag{9}
\]

The spanwise variations have been fitted using connected straight lines. For the deviation variation of figure 2, the correction is averaged using equation (10). According to this piecewise discretization, the numerator and the denominator
reduce to a sum of integrals performed with the help of a symbolic solver.

\[ \int_{r_{	ext{hub}}}^{r_{	ext{tip}}} \Delta \delta(\theta) V_{x(\theta)} r dr = \int_{r_{	ext{hub}}}^{r_{	ext{tip}}} V_{x(\theta)} r dr \]

The remaining expressions of loss and deviation corrections for rotors and stators have been similarly derived, yielding a complete functional correlation for 3D effects. The required boundary layer displacement thicknesses originate from the end-wall boundary layer model, as will be shown later.

The equations (4) to (10) detail the components of the method of velocity triangles. They are combined into a global computation subroutine, along with the necessary correlations. As a result, the aerodynamic and thermodynamic states of the fluid at the upstream and downstream stations of the blade row (see figure 1) are completely defined.

**Blade force source term**

The forces applied by the blade onto the working fluid can be deduced from the knowledge of the velocity triangles.

An angular momentum balance over the blade row supplies the tangential component of the blade force:

\[ r F_{b,\theta} = \frac{q_m}{\text{Vol}} (r_2 V_{\theta,2} - r_1 V_{\theta,1}) \]

In the case of a rotor, this force displaces its application point and is responsible for the shaft work:

\[ SW = \Omega r F_{b,\theta} \]

It is considered that the force exerted by the blade on the fluid is applied in an isentropic way; the viscous effects are applied separately, through the “friction” source term. The blade force is therefore supposed perpendicular to the local relative speed, as shown in figure 4.

\[ \vec{F}_b \cdot \vec{W} = 0 \]

This directly defines the axial component of the blade force:

\[ F_{b,x} = -F_{b,\theta} \frac{W_{\theta}}{V_x} \] (14)

![Figure 4: Blade force and relative speed](image)

Gathering equations (11), (12) and (14), the components of the blade force source term are fully defined:

\[ Q_b = \begin{bmatrix} 0 \\ F_{b,x} S \\ F_{b,\theta} S \\ F_{b,\theta} \Omega r S \end{bmatrix} \] (15)

**Friction source term**

As the blade force application is isentropic, the entropy modification is devoted to a separate equivalent viscous force, according to a distributed loss model, such as proposed in [19].

The magnitude of the dissipative force is related to the entropy creation along a streamline using the definition of the rothalpy and the Gibbs relation [20]:

\[ \left| \vec{F}_f \right| = \rho T \frac{V_m}{W} \frac{\partial s}{\partial m} \]

The curvilinear entropy gradient arises directly from the combined loss coefficient of the velocity triangle computation.

The dissipative force is aligned with the relative speed vector, with an opposite direction.

\[ \vec{F}_f = -\left| \vec{F}_f \right| \frac{\vec{W}}{W} \]

(17)

Once again, only the tangential component of the force is likely to modify the energy balance.
Equations (16) and (17) can be used to determine the complete friction source term:

\[
Q_f = \begin{bmatrix}
0 \\
F_{f,x}S \\
F_{f,\theta}S \\
F_{f,\theta} \Omega r S
\end{bmatrix}
\]  

(18)

**Geometrical source term**

This contribution is independent from the presence of blades within the flow and is related to the shape of the compressor flow path. In the case of a nozzle duct, the mean radius is zero and the only source term is due to cross-sectional surface variations:

\[
\left[ Q_g \right]_{mx} = p \frac{\partial S}{\partial x}
\]  

(19)

For a compressor duct, the mean radius is generally not constant. In the case of important mean radius variations, it has been observed that the angular momentum is not conserved if the tangential equation of (1) is applied as is. Rewriting this equation by conserving the angular momentum over a duct elemental volume, one gets

\[
\frac{\partial (\rho V_\theta S)}{\partial t} + \frac{\partial (\rho V_z V_\theta S)}{\partial x} = -\rho V_z V_\theta S \frac{\partial r}{\partial x} \frac{r}{F_{k,\theta}}
\]  

(20)

This source term can be interpreted as a kinetic force that keeps the flow upon the compressor mean line. Equation (20) contains only the tangential component of this kinetic force.

The direction of this kinetic force is determined so as to avoid creating entropy. The axial counterpart is deduced by considering that the kinetic force is perpendicular to the relative speed.

\[
\vec{F}_k \cdot \vec{W} = 0
\]  

(21)

\[
F_{k,x} = -\rho V_\theta W_\theta S \frac{\partial r}{r} \frac{r}{\partial x}
\]  

(22)

Only the tangential component participates in the energy balance. The final form of the geometrical source term is

\[
Q_g = \begin{bmatrix}
0 \\
F_{k,x} S + p \frac{\partial S}{\partial x} \\
F_{k,\theta} S \\
F_{k,\theta} \Omega r S
\end{bmatrix}
\]  

(23)

This fictive kinetic force ensures that the angular momentum is conserved in non-bladed zones, respects the Euler law in the bladed zones, and does not modify the fluid entropy.

**EQUATION SOLVER**

The complete set of equations to be solved gathers relations (1), (3), (15), (18) and (23). It describes a two-dimensional flow, developing on an axisymmetric flow surface generated by the revolution of the compressor mean line. This “quasi-1.5D” version of the Euler equations can be solved by classical finite volume algorithms, with a minimum of adjustments.

The computation domain is the real compressor flow path, discretized by a one-dimensional structured mesh, as shown in figure 5. It can be observed that the blade rows are divided internally into control volumes, in order to obtain moderate flow deflections for each elementary cell.

**Figure 5: One-dimensional compressor mesh**

On each control volume, the integral form of the Euler equations is written, resulting in a relatively large linear system of equations. This formulation makes use of upwind discretization of the fluxes to ensure a good stability of the computation. Roe’s flux difference splitting is used throughout the test cases presented here. The resulting flow solver is representative of the most up-to-date CFD techniques, particularized to the quasi-one-dimensional flows observed in axial compressors. It is ready for unsteady computations and offers a wide application range, thanks to the generic approach of technological effects brought by the source terms.

The computer code handles conservative variables located at the cell centers. A quadratic, limited, hybrid reconstruction scheme extrapolates the discrete solution across the domain when needed [21][22].

The boundary conditions applied at the inlet and outlet of the compressor, for an unstalled forward flow, are as follows:

<table>
<thead>
<tr>
<th>Inlet</th>
<th>Outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>imposed</td>
<td>$p_{in}$, $T_{in}$, $\alpha_{in}$</td>
</tr>
</tbody>
</table>

**Table 1: Boundary conditions**
For a steady-state simulation, a time-marching solver allows the system to evolve towards a stationary solution. The associated time integration is performed by implicit or explicit schemes.

The explicit scheme is based on a four-step Runge-Kutta time integration; the stationary mode uses coefficients which maximize the time dissipation.

The implicit scheme works with an Euler single-step forward time differentiation. For the stationary version, the system of equations is linearized and solved in one iteration at each time step. A smart node-coloring finite-difference scheme defines each Jacobian matrix. It takes advantage of the reconstruction method and of the mesh structured aspect to limit the computational burden.

For steady-state compressor simulations, the implicit scheme is undoubtedly the fastest option. Furthermore, it has proved to be more stable for problematic operating conditions. But the explicit scheme may come in handy for transient or unsteady simulations, when the physical time step is too small to justify the larger cost of implicit time iterations [23].

**ANNULUS BOUNDARY LAYER MODEL**

A realistic model for axial compressor operation requires the simulation of the annulus boundary layers.

The simple model proposed by Stratford [24] has been chosen. It considers the overall changes in boundary layer axial displacement thickness across the successive blade rows. The losses associated with the end-wall boundary layers and tip clearances are already rendered by the averaged “3D” corrections to the overall loss coefficient. The boundary layer model just has to produce an estimate of the displacement thicknesses. The blockage due to end-wall boundary layers is then integrated into the cross-sectional areas appearing in the quasi-1.5D Euler equations (1).

The Stratford model solves an axial momentum balance across the compressor. The boundary layer shape factor and skin friction are roughly estimated. The force defect is assumed to be zero and the boundary layer flow is supposed collateral. These approximations yield a fair estimate of axial blockage [25]. The absence of information about the losses or cross-flows does not pose a problem in this case. Besides, the “3D” loss contribution based on the results of Roberts et al uses the obtained displacement thickness to predict these losses.

The annulus boundary model is applied at each time step, based on the current solution. The implicit scheme does not integrate it into the Jacobian matrix estimation to limit the computation effort.

**APPLICATIONS**

**Single subsonic compressor stage**

The first simulation example is an open literature test-case representative of a civil high-pressure compressor [26]. It consists of a single, close-coupled stage with moderate loading. The blading does not present any “end-bends”; the blade profiles are stacked with their centroids on a radial line.

The rotor blade profiles are of DCA profile while the stators have a C4 mod 2 thickness distribution on a circular-arc camberline. Both types are conventional and the open literature correlations can reasonably be expected to predict accurately their losses and deviations.

A simplified version of the one-dimensional mesh applied to the compressor flow path is shown in figure 6. The real mesh counts more than one hundred mesh cells and is less readable. The simulation may run on coarser meshes but the reduced execution time makes it possible to work with a relatively large number of cells, thus decreasing the discretization error. Note that the blade row leading and trailing edge projections must match cell boundaries inside the computation domain.

![Figure 6: Single-stage compressor flow path mesh](image)

The whole compressor map is available; the simulation has been run to cover the same operational domain, ranging from 70% to 110% of the nominal rotation speed.

The total pressure ratio and isentropic efficiency are given as functions of the real mass flow in figures 7 and 8. The program options used are listed below.

- Reference incidence: NASA 2D, according to Lieblein [9]
- Reference deviation : Carter’s law
- Off-design deviation: Creveling’s relation
- Primary losses: Creveling’s relation [10]
- Secondary losses: equation (8)
- 3D correction for losses and deviations
- Stratford’s end-wall boundary layer model
Three-stage axial compressor

The compressor chosen as a second validation example is a test model for the last three stages of a civil eight-stage compressor [27], subsequently named PW3S1.

The flow path is characterized by a constant mean radius, and a constant cross-sectional area. A glimpse of the simple mesh created for this geometry is presented in figure 9. Once again, the real mesh would fail to clearly show the cell distribution, since it contains about 200 control volumes.

Finally, figure 7 shows that the annulus boundary layer model is far from perfect, as it fails to render the choking at high mass flows and high rotation speeds. A parametric identification could be used to optimize the model of Stratford, along with the associated losses. This optimization process should also be used to compensate for missing measured data, as accurate values for the inlet boundary layer thicknesses and outlet static pressures.

Figure 8 shows a general trend of the model to over-estimate the isentropic efficiency, even if in some parts of the map the losses are clearly too important. The peak efficiencies are however correctly simulated. The observed constant error could then be reduced by using a more advanced secondary loss model.

The correlation tuning process will instead be demonstrated on the following test-case, owing to its unconventional blade profiles.

Three-stage axial compressor

The compressor chosen as a second validation example is a test model for the last three stages of a civil eight-stage compressor [27], subsequently named PW3S1.

The flow path is characterized by a constant mean radius, and a constant cross-sectional area. A glimpse of the simple mesh created for this geometry is presented in figure 9. Once again, the real mesh would fail to clearly show the cell distribution, since it contains about 200 control volumes.

Finally, figure 7 shows that the annulus boundary layer model is far from perfect, as it fails to render the choking at high mass flows and high rotation speeds. A parametric identification could be used to optimize the model of Stratford, along with the associated losses. This optimization process should also be used to compensate for missing measured data, as accurate values for the inlet boundary layer thicknesses and outlet static pressures.

Figure 8 shows a general trend of the model to over-estimate the isentropic efficiency, even if in some parts of the map the losses are clearly too important. The peak efficiencies are however correctly simulated. The observed constant error could then be reduced by using a more advanced secondary loss model.

The correlation tuning process will instead be demonstrated on the following test-case, owing to its unconventional blade profiles.
Figure 11 indeed illustrates the accuracy of the efficiency prediction, showing only a constant offset that can easily be compensated by tuning the secondary losses equation.

Figure 11: “Standard correlations” efficiency line

Consequently, a parametric identification procedure has been conducted to demonstrate the generalization capabilities of the Quads code. Some parameters thus modify the standard correlations of the CFD code.

The objective of this procedure is to identify the parameters on the sole basis of the 100% nominal speed data. It is hoped that the modified loss and deviation models will be able to provide a good solution in an extended range of mass flows and rotation speeds.

The choice of optimization parameters is explained hereafter.

- Off-design deviations and primary losses

Figure 10 suggests that the reference incidence, deviation and primary loss coefficient are fairly defined by the standard correlations. Some tuning coefficients are then introduced into the off-design deviation angle and primary loss correlations.

The formulation of Creveling, equation (6), has been kept for the off-design deviation angle. But the expression of the \( f \) function has been replaced by two distinct cubic polynomials that match at \((0,0)\).

\[
x = \frac{i - i^*}{\Delta \beta^*}
\]

\[
\begin{align}
x > 0 & \Rightarrow f(x) = c_3x^3 + c_2x^2 + c_1x \\
x < 0 & \Rightarrow f(x) = c_3x^3 + c_2x^2 + c_1x
\end{align}
\]

The primary loss coefficient also keeps the form used in Creveling’s relation (7). However, the numerous simulations conducted on the PW3S1 compressor show that the relative Mach number is relatively constant over the considered operation interval, at nominal rotation speed. It was then chosen to introduce only two parameters in the expression of the off-design primary loss coefficient:

\[
\begin{align}
(i - i^*) > 0 & \Rightarrow \sigma - \sigma^* = c_{m,p} \cdot (i - i^*)^2 \\
(i - i^*) < 0 & \Rightarrow \sigma - \sigma^* = c_{m,n} \cdot (i - i^*)^2
\end{align}
\]

- Boundary layer displacement thickness

The boundary layer thickness at the compressor inlet is not specified in the published test data [28] and it is therefore included in the set of optimization coefficients. A unique coefficient is introduced into the annulus boundary layer model. It represents the ratio of the wall boundary layer thickness to the channel height, at the inlet of the compressor.

- Secondary losses

Finally, a tuning coefficient is added to the secondary losses equation, in order to compensate for the offset efficiencies of figure 11. A parameter is therefore incorporated in the expression of the secondary losses coefficient:

\[
\sigma_s = k_{sl} \cdot L_{sl} \cdot \sigma \cdot \frac{\cos^2 \beta}{\cos^2 \beta_w}
\]

The parametric identification problem has been solved by using a Nelder-Mead optimization algorithm [29]. It is a “0-order” method and does not need the objective function derivatives. The algorithm was chosen because of its good global minimum finding capabilities and the linearly increasing computational burden with the number of variables.

The objective function is defined as a weighted sum of the relative errors committed on the predicted mass flow, pressure ratio and efficiency. After reaching an acceptable solution, the optimized correlations permit a close prediction of the PW3S1 compressor performance at 100% nominal speed. These correlations were used to simulate once again the whole compressor map. The results are shown in figures 12 and 13.

The aforementioned optimization processes have been made possible thanks to the reduced execution time of the analysis tool. A typical run needs as
low as one second CPU on a personal computer, including geometry input, first guess computation, implicit time-marching resolution of the quasi-1.5D equations, and output data treatment.

With eight coefficients, the optimization process needs about one thousand objective function estimations to reach a worthwhile optimum. Each objective function needs the simulation of about five operation points. Full parameter identification consequently needs one to two hours to complete, which allowed a thorough exploration of the code capabilities.

With eight coefficients, the optimization process needs about one thousand objective function estimations to reach a worthwhile optimum. Each objective function needs the simulation of about five operation points. Full parameter identification consequently needs one to two hours to complete, which allowed a thorough exploration of the code capabilities.

The adapted off-design deviation angle correlation correctly renders the pressure ratio collapsing at low mass flows. On the other hand, at high mass flows, the deviation angle correlation of Creveling also had to be modified to match the experimental data.

The 85%NN and 105%NN operating curves show the good generalization capabilities of the “nominal” optimized correlations, for the pressure ratio as well as for the isentropic efficiency.

Figure 12: “Optimized correlations” pressure lines

The 85%NN and 105%NN operating curves show the good generalization capabilities of the “nominal” optimized correlations, for the pressure ratio as well as for the isentropic efficiency.

The adapted off-design deviation angle correlation correctly renders the pressure ratio collapsing at low mass flows. On the other hand, at high mass flows, the deviation angle correlation of Creveling also had to be modified to match the experimental data.

The 85%NN and 105%NN operating curves show the good generalization capabilities of the “nominal” optimized correlations, for the pressure ratio as well as for the isentropic efficiency.

The adapted off-design deviation angle correlation correctly renders the pressure ratio collapsing at low mass flows. On the other hand, at high mass flows, the deviation angle correlation of Creveling also had to be modified to match the experimental data.

The adapted off-design deviation angle correlation correctly renders the pressure ratio collapsing at low mass flows. On the other hand, at high mass flows, the deviation angle correlation of Creveling also had to be modified to match the experimental data.

Figure 13: “Optimized correlations” efficiency lines

The adapted off-design deviation angle correlation correctly renders the pressure ratio collapsing at low mass flows. On the other hand, at high mass flows, the deviation angle correlation of Creveling also had to be modified to match the experimental data.

The 85%NN and 105%NN operating curves show the good generalization capabilities of the “nominal” optimized correlations, for the pressure ratio as well as for the isentropic efficiency.

Figure 14: Off-design deviation transfer function

The optimized primary loss correlation can be similarly analyzed. It predicts higher losses in the positive incidence difference zone, and lower losses in the negative zone, than those given by the equation of Creveling.
CONCLUSIONS

A quasi-one dimensional model for axial compressors has been presented. The simulation tool is introduced as the basis of a complex model for a wide range of axial compressor configurations and operations.

The correlations introducing empirical information into the simulation originate from the open literature. As the situation requires, they can be exchanged for a manufacturer’s own correlations, or tuned by means of optimization coefficients.

The two application examples illustrate these possibilities. The analysis tool has been used to explore the characteristic map of given compressors. Equipped with adequate correlations, the developed computer code has shown to simulate their behaviour with excellent agreement when compared to measured data.

The introduction of physical effects through source terms is a very generic approach and could be used to simulate more complex configurations or effects. The model could thus be extended to combustion chambers [30] and turbines, to simulate the operation of a whole gas turbine engine [23]. Water ingestion, blade fouling or cooling devices may also be introduced.

The selected model gives a wide application range to the simulation. The applicability ranges from stationary to unstationary, from low speed to high speed, from forward to reverse flows. In the near future, further developments will be directed towards stall detection, which appears necessary in the scope of a useful performance prediction tool [16]. The extrapolation of compressor maps to very low speeds is also planned, to study the windmilling of compressors.

REFERENCES


