A FINITE ELEMENT PARALLEL DOMAIN DECOMPOSITION METHOD FOR INCOMPRESSIBLE TURBOMACHINERY FLOWS

Domenico Borello*, Alessandro Corsini*, and Franco Rispoli*

* Dipartimento di Meccanica ed Aeronautica
  Università di Roma “La Sapienza”
  Via Eudossiana 18, I00184 Roma, Italy
  e-mail: rispoli@dma.ing.uniroma1.it, web page: http://dma.ing.uniroma1.it/

Key words: Turbomachinery Flows, Turbulence Modelling, Stabilized Finite Elements, Domain Decomposition, Parallel Processing.

Abstract. The paper presents a domain decomposition algorithm applied to a finite element code to model steady, incompressible, turbulent internal fluid flows. The domain decomposition parallelization is here considered as a key feature for the simulation of complex flow phenomena in turbomachinery where a huge number of computational grid points is necessary. An eddy viscosity modelling is adopted for the fluid dynamic closure. The Reynolds stress tensor terms are modelled by both linear and non-linear cubic versions of k-ε high Reynolds number turbulence model. The finite element formulation is based on a stabilized Petrov Galerkin method modified for the application to tri-linear and tri-quadratic elements in three-dimensions. A residual relaxation of the incompressibility constraint is adopted in order to circumvent the Babuska-Brezzi condition for (Q1-Q1) elements, whereas within a mixed finite element formulation (Q2-Q1) it permits to improve the convergence behavior of adopted iterative solver. As far as the domain decomposition implementation is concerned, the proposed methodology is based on an additive Schwarz overlapping domain decomposition approach applied to matching grids. The solution strategy combines such a domain decomposition technique with a block Gauss-Siedel preconditioner and a GMRes solver. The computational storage is further reduced by adopting a compact skyline addressing of stiffness matrix and a grid renumbering algorithm. To demonstrate the efficiency of the proposed method two and three dimensional benchmark flows are computed on both the SP2 IBM and CRAY T3E parallel machines. Finally, an analysis of two and three dimensional turbulent flows arising in turbomachines will be carried out. Discussion of numerical results and comparison with experimental data will conclude this work.
1 INTRODUCTION

High performance rotors and narrow design margins for future turbomachinery demand accurate flow analysis methods. With the advance of computational power, numerical methods that solve the Reynolds averaged Navier-Stokes equations not only provide detailed flow analyses but become an important element of turbomachinery design. The highly complex physics of the flow field structure arising in turbomachines, requires the development of increasingly accurate simulation models, able to adequately evaluate the influence of viscous and three-dimensional (3D) effects. In most cases such needs could be faced allowing the concentration of large share of computational grids to model flow regions where end-wall and clearance phenomena, losses and unsteady effects occur. Thus a high-performance computing (HPC) platform is mandatory. For such reasons, an effective parallel domain decomposition (DD) computational strategy is here proposed, on the basis of the in-house developed finite element Navier-Stokes solver\(^1,2\), which overcomes the limits inherent in so called standard computational approach in terms of both flow modelling and computational algorithm.

From the modelling viewpoint, the proposed methodology features, in addition to standard two-equations high Reynolds number eddy viscosity model (EVM), a non-linear cubic \(k\)-\(\varepsilon\) model which allows recovering the anisotropy of the Reynolds stress tensor to a good degree. Therefore it establishes a sensitivity to the effects of strong streamlines curvature and rotating turbulence behaviour\(^3\), featuring turbomachinery flows. The adopted non-linear \(k\)-\(\varepsilon\) model\(^4\) has been the subject of intensive validation efforts in several flow problems (such as: external aerodynamics, environmental problems, etc.). With few exception\(^2,5\) it has not, however, been investigated in the context of turbomachine flows where simpler EVM are traditionally applied.

From the computational viewpoint, the proposed parallel DD method has been developed within FEM due to its ability to model fluid-dynamic problems with complex domains, a critical benefit in the turbomachinery CFD. Both mixed (Q2-Q1) and equal order (Q1-Q1) interpolation spaces have been used in the FEM approximation methodology, and numerical instabilities are faced through the adoption of strongly consistent Petrov-Galerkin formulation\(^6,7,8\). The guaranteed stability-accuracy compromise represents a key-feature of such a numerical method versus the standard ones, as shown by the comparison against a Finite Volume (FV) formulation in the simulation of simple two-dimensional (2D) inviscid benchmark flow. The developed parallel solution methodology is based on an Additive Schwarz Overlapping Domain Decomposition procedure due to its intrinsic parallelism\(^9,10\). The implemented approach combines a slightly Overlapping Domain Decomposition procedure together with a block Gauss-Siedel preconditioner, and a GMRes solver\(^11,12\). An original solution procedure is here proposed which merges inner Krylov, outer Schwarz and fixed point non linear iterations, leading to accelerated convergence histories. The data decomposition has been carried out using an in-house made code properly developed to guarantee both the minimization of the message passing requirements and the load balancing.
In order to reduce the storage requirements for the stiffness matrix, both a compact skyline algorithm\textsuperscript{13}, with addressing vector\textsuperscript{2}, and a renumbering algorithm\textsuperscript{14} has been implemented. For the message passing operations the MPI libraries have been used\textsuperscript{15}.

The proposed solution strategy has been validated against 2D and 3D benchmarks, on both the SP2 IBM and Cray-T3E parallel computers. The first analysis has concerned the performance of the parallel solver by evaluating the scalability parameters for laminar and turbulent benchmarks dealing with strongly distorted flows (2D laminar flow in a backward-facing step and turbulent flow through 90° bend, 3D laminar flow in curved duct). The second analysis is focused on highly challenging turbomachinery flows (DCA compressor cascade and 3D isolated non-free vortex fan). The comparison with the experimental data has been carried out pointing out the ability of the proposed parallel code to predict the main flow phenomena, underlining likewise the limits of the turbulent modelling here adopted.

2 FLUID MODEL FORMULATIONS

In the present paper, the physics involved in the fluid dynamics of incompressible turbulent flows, in rotating and stationary frames of reference, is modelled by the averaged Navier-Stokes equations. The Reynolds decomposition has been adopted, so that each quantity $U$ is decomposed into its conventional average value (indicated by the overbar symbol) and its fluctuation with respect to the latter (denoted by the prime symbol), $U = \overline{U} + U'$. Two EVMs have been implemented in the proposed solution algorithm. First the standard $k$-$\varepsilon$ model\textsuperscript{16} which adopts a Newtonian-like eddy-viscosity concept. Then, the non-linear $k$-$\varepsilon$ model\textsuperscript{4} which instead adopts an anisotropic constitutive relation, in the form of a third-order polynomial in strain and vorticity tensors, able to recover sufficient variety in the stress-strain couplings.

2.1 Problem statement

The dynamic response of incompressible turbulent fluids is described by the boundary value problem expressed for the following set of fluid quantities (momentum components $\rho u_i$, ($i=1,2,3$) (where $\rho$ is the density of the fluid assumed constant, and $u$ the Cartesian velocity components), pressure $p$, turbulent kinetic energy $k$ and viscous dissipation rate $\varepsilon$), in general form

\begin{equation}
F_{a,j} + F_{d,j} + \rho B = 0 \quad \text{in } \Omega, \text{ for } j = 1,2,3 \tag{1}
\end{equation}

\begin{align*}
U &= \Phi_D \quad \text{on } \partial \Omega_D \\
F_{d,n} &= \Phi_N \quad \text{on } \partial \Omega_N
\end{align*}

$U$ is the fluid averaged local unknowns vector defined as

\begin{equation}
U \equiv [\overline{u}_1, \overline{u}_2, \overline{u}_3, \overline{p}, k, \varepsilon]^T = \overline{U} + [0, 0, 0, \overline{p} - 1, 0, 0]^T \tag{2}
\end{equation}

$F_a$, $F_d$ are the advective and the diffusive fluxes respectively, depending on the density, the
corrected unknowns vector $\bar{U}$ and its gradient as

$$F_d\left(\bar{U}, \nabla \bar{U}\right) = \begin{bmatrix} u_{ij} \rho \bar{u}_1, & u_{ij} \rho \bar{u}_2, & \bar{u}_{ij} \rho, & \bar{u}_{ij} \rho k, & \bar{u}_{ij} \rho e \end{bmatrix}^T$$

(3a)

$$F_d\left(\bar{U}, \nabla \bar{U}\right) = \begin{bmatrix} \tau_{ij}, & \tau_{2j}, & \tau_{3j}, & 0, & -p\left(\nu + \frac{v_t}{\sigma_k}\right) k_{ij}, & -p\left(\nu + \frac{v_t}{\sigma_e}\right) e_{ij} \end{bmatrix}^T$$

(3b)

where the deviatoric stress tensor and the corrected pressure are defined as

$$\tau_{ij} = \bar{p}^* \delta_{ij} - p\left(\nu + \frac{v_t}{\sigma_k}\right) \left(u_{ij} + \bar{u}_{ij}\right), \quad \bar{p}^* = \bar{p} + (2/3) \rho k$$

(4)

In (3b) $v_t$ is scalar kinematic turbulent viscosity, $\nu$ is the molecular viscosity and the Prandtl empirical numbers are taken as $\sigma_k = 1, \sigma_e = 1.3$. The source vector $B$ is given by

$$B = \begin{bmatrix} f_{in1} + P_{M1}^{(nl)} , & f_{in2} + P_{M2}^{(nl)} , & P_{M3}^{(nl)} , & 0, & -P_k + \nu , & -c_{e1} \frac{\nu}{k} P_k + c_{e2} \frac{\nu^2}{k} \end{bmatrix}^T$$

(5)

Here the forces applied to momentum components concern two vector contributions. The first one, $f_{in}$, takes into account the effects of non-inertial frame of reference

$$f_{in} = \left[ -\omega x_1 - 2\omega \bar{u}_2, \quad -\omega x_2 + 2\omega \bar{u}_1, \quad 0 \right]^T$$

(6)

the terms of that vector originate from a uniform anti-clockwise rotation of rate $\omega$ about positive $x_3$ – direction. While the second vector contribution $P_{M}^{(nl)}$ deals with the gradient of both second and third-order terms appearing in non-linear Reynolds stress constitutive relation$^{17}$.

$$P_{M}^{(nl)} = \sum_j \left[ c_{\nu1} \frac{k}{\nu} \left( S_{ik} S_{kj} + S_{kl} S_{li} \frac{I}{3} \delta_{ij} \right) + c_{\nu2} \frac{k}{\nu} \left( W_{ik} S_{kj} + W_{jk} S_{ki} \right) + c_{\nu3} \frac{k}{\nu} \left( W_{ik} W_{kj} - W_{kl} W_{kl} \frac{I}{3} \delta_{ij} \right) + c_{\nu4} c_{\mu} v_t \left( \frac{k}{\nu} \right)^2 \left( S_{ik} W_{kj} + S_{kj} W_{li} - S_{km} W_{lm} \frac{2}{3} \delta_{ij} \right) + \right] S_{kl}$$

$$+ c_{\nu5} c_{\mu} v_t \left( \frac{k}{\nu} \right)^2 \left( S_{ik} S_{kj} - S_{km} S_{lm} \frac{I}{3} \delta_{ij} \right) S_{kl} +$$

$$+ c_{\nu6} c_{\mu} v_t \left( \frac{k}{\nu} \right)^2 \left( S_{ij} S_{kl} S_{kl} + c_{\gamma} c_{\mu} v_t \left( \frac{k}{\nu} \right)^2 \right) S_{ij} W_{kl} W_{kl} \right] j$$

(7)

Under the hypothesis of high Reynolds formulation, the turbulent viscosity is

$$\nu = c_{\mu} \left( k^2 / \nu \right)$$

(8)
where \(c_\mu\) for standard and non-linear EVM is respectively defined as

\[
c^{(s)}_\mu = 0.09 \quad \quad c^{(nl)}_\mu = \frac{0.3 \cdot \left[1 - \exp\left(-0.36 / \exp\left(-0.75 \max(S, W)\right)\right)\right]}{1 + 0.35 \max(S, W)}
\]  

while \(S\) and \(W\) represent respectively the strain and the absolute vorticity invariants

\[
S = \frac{k}{\varepsilon} \sqrt{\frac{1}{2} S_{ij} S_{ij}} \quad \quad W = \frac{k}{\varepsilon} \sqrt{\frac{1}{2} W_{ij} W_{ij}}
\]

The values of constants in (7) are taken as

<table>
<thead>
<tr>
<th>(c_1)</th>
<th>(c_2)</th>
<th>(c_3)</th>
<th>(c_4)</th>
<th>(c_5)</th>
<th>(c_6)</th>
<th>(c_7)</th>
</tr>
</thead>
<tbody>
<tr>
<td>-0.1</td>
<td>0.1</td>
<td>0.26</td>
<td>-1.0</td>
<td>0.0</td>
<td>-0.1</td>
<td>0.1</td>
</tr>
</tbody>
</table>

Table 1: Non-linear EVM constants

As far as the turbulent production term \(P_k\) is concerned it is made proportional to the gradient of Reynolds stress tensor. No modifications have been introduced in order to tune the turbulence modelling constants, therefore the following standard values are taken

\[
\sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3, \quad c_{\varepsilon 1} = 1.44, \quad c_{\varepsilon 2} = 1.92
\]

The set of boundary conditions imposed on the integration domain boundary, implies the application of inflow Dirichlet in terms of velocity profiles and turbulent variables distributions, as well as outflow natural Neumann conditions except from the case of non-inertial frame of reference when non-homogeneous stress flux is applied. Synthetic boundary conditions are enforced in the near wall regions in order to simulate the steep gradients taking place there. Such conditions simulate the wall shear stress and impermeability, a Neumann like condition for \(k\) according to the localization of the near-wall peak within the sublayer\(^{18}\), and a Dirichlet \(\varepsilon\) value spreading out from the balance between turbulent production and dissipation.

Finally, the non-linearity of the system is solved by the robust fixed point technique which transforms the original incomplete parabolic form of equations (1) into a generalized Oseen problem. Such an approach leads to a Stokes like problem formulation, where the first order terms in (1) are multiplied by a given vector field (the velocity fields itself evaluated at the preceding equilibrium iteration).

### 3 STABILIZED FINITE ELEMENT FORMULATION

The numerical solution of the boundary value problem presented above, is approached by the use of a FEM discretization based on a Petrov-Galerkin (PG) weighted residual method allowing the use of test function spaces different from the trials one and not necessarily continuous. The adopted Petrov-Galerkin formulation is addressed to control the main instability origins, that affect incompressible flows numerical prediction, \textit{via} the perturbation of the original Galerkin weighting function set. The original sets of symmetric weighting...
functions, is perturbed in order to introduce streamwise artificial diffusive balancing terms in
the model advective-diffusive equations (Streamline Upwind/PG (SU/PG) like terms\(^6\)), and a
relaxation of incompressibility constraint Laplacian like in the continuity equation (Pressure
Stabilized (PS/PG) like terms\(^7,2\)). The adoption of a PS/PG like stabilization is able to
circumvent the Babuska-Brezzi condition for (Q1-Q1) elements, whereas, within a mixed
finite element formulation (Q2-Q1) the lack of zero diagonal entries permits to improve the
conditioning characteristics of non-linear finite element matrix.

The vector of the unknowns could be then defined as: \(U = U_p + U_c\), where
\(U_p = \begin{bmatrix} u_1, u_2, u_3, 0, k, e \end{bmatrix}^T\) is the vector of the primary-turbulent flow properties (quadratic for
Q2-Q1 element or linear for Q1-Q1 element), while \(U_c = \begin{bmatrix} 0, 0, 0, \tilde{p}^*, 0, 0 \end{bmatrix}^T\) deals with the linear
constraint variable.

Given a finite element partition of the original closed domain \(\Omega\) into elements \(\Omega_e, e = 1, nel\)
\((nel\) number of elements) such that
\[
\bigcup_\varepsilon \Omega_e = \Omega \quad \text{and} \quad \bigcap_\varepsilon \Omega_e = \emptyset
\]  

Consider the definition of interior boundary as
\[
\partial \Omega_{int} = \bigcup_\varepsilon \partial \Omega_e / \partial \Omega
\]  

\(\partial \Omega_{e}\) denotes \(\Omega_{e}\) boundary and \(\partial \Omega = \partial \Omega_N \cup \partial \Omega_D\) is the computational domain boundary of
\(\Omega\) (where \(\partial \Omega_D\) and \(\partial \Omega_N\) are open disjoint subsets of \(\partial \Omega\)). Let define, the finite dimensional spaces of trial functions as
\[
S_p^h = \left\{ j^h_p | U_p^h \in H^{1h}(\Omega), U_p^h = \phi_D \quad \text{on} \quad \partial \Omega_D, \quad \phi_D \in H^{1/2h}(\partial \Omega_D) \right\}
\] 

\[
V_p^h = \left\{ w_p^h | w_p^h \in H^{1h}_0(\Omega), \quad w_p^h = 0 \quad \text{on} \quad \partial \Omega_D \right\}
\] 

\[
S_c^h = V_c^h = \left\{ j^h_c | U_c^h \in H^{1h}_0(\Omega), \quad w_c^h = 0 \quad \text{on} \quad \partial \Omega_D \right\}
\]  

here \(H^{1h}(\Omega), \quad H^{1/2h}_0(\Omega)\) are the Sobolev spaces for the continuous pair of finite element
defunctions, \(H^{1/2h}(\Omega)\) is the restriction to the domain boundaries, and the superscript \(h\) refers
to the characteristic length scale of the domain discretization. The associated finite element
weighting functions result from the composition of continuous Galerkin weights \((w_p^h \in V_p^h, \quad w_c^h \in V_c^h)\) with perturbation operators, locally proportional respectively to \(w_p^h\) and \(w_c^h\)
gradient \((\tilde{C}^0\) over each elementary domain \(\Omega_e)\)
\[
\pi_p^h = t_p \tilde{u}_j w_p^h, \quad \pi_c^h = t_c w_c^h
\]  

where,
Domenico Borello, Alessandro Corsini, and Franco Rispoli.

\[ t_p = \frac{h}{2\|u\|_2} z(Re_u), \quad t_c = \frac{h}{2\|V\|_2} z(Re_V) \]

are intrinsic time scales, representing transit time for the information to be advected over the element. Here \( Re_u \) and \( Re_V \) are element Reynolds number, based respectively on local velocity \( \|u\|_2 \) and global scaling velocity \( \|V\|_2 \) to assure a non-zero stabilization in local diffusion dominated flow regions. The non-dimensional numerical diffusivity \( z(Re) \) is the so-called magic function, used in its critical form\(^{19}\) to control the stabilization local Stokes limit.

Thus the modified weights may be written in vector form as: \( w^h = [w_p, w_p, w_p, w_p, w_p]^T, \pi^h = [\pi_p, \pi_p, \pi_p, \pi_p, \pi_p]^T \) (15)

The stabilized Petrov-Galerkin formulation of differential problem (1) reads

\[
\text{find } U_p^h \in S_p^h, \quad U_c^h \in S_c^h \quad \forall w_p^h \in V_p^h, \quad \forall w_c^h \in V_c^h, \text{ such that }
\]

\[
c(r^h; U^h, w^h) + s(U^h, w^h) + \Pi(\pi^h) = (\rho B^h, w^h) + (\phi_N, w_p^h \phi_{\Omega_N})_{\partial \Omega_N}
\]

with use of bi-linear and tri-linear forms \( \forall U_p^h \in S_p^h, \quad U_c^h \in S_c^h \) and \( \forall w_p^h \in V_p^h, \quad w_c^h \in V_c^h \)

\[
s(U^h, w^h) = \int_{\Omega} w_j^h : F_{dj}^h d\Omega
\]

\[
\left(\rho B^h, w^h\right) = \int_{\Omega} w^h \rho B^h d\Omega
\]

\[
\left(\phi_N, w_p^h \phi_{\Omega_N}\right)_{\partial \Omega_N} = \int_{\partial \Omega} w_p^h \phi_N d(\partial \Omega)
\]

\[
c(r^h; U^h, w^h) = \int_{\Omega} \left(\tilde{r}_j^h F_{\alpha i}^h\right)_j w^h d\Omega
\]

The stabilization terms are defined as

\[
\Pi(\pi_p^h) = \lambda_p \sum_{e=1}^{nel} \int_{\Omega_e} \left( F_{a,i}^h + F_{d,i}^h + \rho B^h \right) \pi_p^h d\Omega
\]

\[
\Pi(\pi_c^h) = \lambda_c \sum_{e=1}^{nel} \int_{\Omega_e} \left( F_{a,i}^h + F_{d,i}^h + \rho B^h \right)^M \pi_c^h d\Omega
\]

In (18) the integration of stabilizing contributions is confined on the element interior, according to perturbation function existence domain. Furthermore, the PS/PG stabilization integrals involve only the momentum balance residual (indicated by the superscript (M) in (18)). To ensure the weakly fulfillment of the original set of Neumann conditions\(^{20}\) is required the application of a stability coefficient \( \lambda_c \) able to correct the effect of the artificial boundary
flux perturbations\textsuperscript{21,22}. On the other hand, the coefficient $\lambda_p$ in (18) is a tuning parameter used to achieve the best stability-accuracy compromise for the solution.

In order to formulate the DD scheme, the non-homogeneous value variational problem (16) could be transformed by expliciting the effect of Dirichlet boundary data. In that case, the variational formulation reads

$$\text{find } V^h \in H^h_0 \forall w_p^h \in V_p^h, \forall w_c^h \in V_c^h, \text{ such that}$$

$$c(u^h; V^h, w^h) + s(V^h, w^h) + \Pi \left( \pi^h \right) = \left( \rho B^h, w^h \right) + \left( \varphi_N^h, \tau^h \right)_{\partial \Omega_N} - F(w^h)$$

(19)

where

$$F(w^h) = c(u_D^h; \Phi^h, w^h) + s(\Phi^h, w^h) + \Pi \left( \pi^h \right)_{\Phi_o^h}$$

(20)

defines the integral influence of boundary values on the unknowns $V^h$ carried out by introducing $\Phi_D$ that denotes any extension in $\Omega$ of the non-homogeneous Dirichlet datum. It is worth to note that (20) includes also the integral contribution $\Pi \left( \pi^h \right)_{\Phi_o^h}$ that originates from the consistent stabilization. The original complete solution $U^h$ is thus obtained by adding $V^h$ and $\varphi_D$. The application of a generalized Oseen problem to (16) involves a linearization of the original formulation. The consequent linearized system is here approached by a GMRes solution strategy applied to a compact skyline renumbered matrix\textsuperscript{11,14,17}.

3.1 Evaluation of stabilization effects

In order to show the accuracy of the developed FEM formulations is here carried out a comparison among an artificial dissipation finite volume (FV) formulation\textsuperscript{23}, and two FEM stabilized formulations: a Streamline-Upwind (SU, where the perturbation operator is applied as a balancing diffusion tensor) PS/PG formulation, and a consistent SU/PG-PS/PG scheme. Both equal (Q1-Q1) and mixed (Q2-Q1) order elements are considered. The benchmark was a 2D inviscid flow through a 45° bend, with mean radius to duct width ratio equal to 5.5 and the straight inlet and outlet lengths each equal two duct widths long. As far as the discretization in concerned, 53 grid points are uniformly distributed along the stream direction and 15 grid points are adequately stretched crosswise. A uniform inlet velocity profile is assumed. The number of GMRes basis vectors is set equal to 50. The number of non-linear iterations for all combinations here presented are such that the log of relative residual $r$ norm ($R_{res} = \|r_k\|_2/\|\tilde{r}_0\|_2$) and the difference between two consecutive solution norms ($R_{sol} = \|U_k\|_2 - \|U_{k-1}\|_2/\|U_k\|_2$ of equation system derived from (18) are both less than -6.

The global parameters, proposed by Basson and Lakshminarayana\textsuperscript{23}, and here adopted for the comparative estimate are: the difference between the inlet ($\hat{m}_{in}$) and outlet ($\hat{m}_{out}$) mass flow rates, normalized by the inlet mass flow rate; the difference between inlet and outlet
mass averaged total pressures \((P_{\text{in}}^{\text{av}}\) and \(P_{\text{out}}^{\text{av}}\)), normalized by the inlet dynamic head \(Q_{\text{in}}\); the difference between the maximum and minimum total pressures \((P_{\text{in}}^{\text{max}}\) and \(P_{\text{in}}^{\text{min}}\) respectively) in the overall field, normalized by the inlet dynamic head. The estimation parameters are all expressed as a percentage of the inlet reference quantities. The data in Table 2 clearly show the capability of FEM based stabilized formulations to strongly preserve the consistency of the solutions also in a pure advective limit, without introducing high amount of numerical disturbance. It is furthermore interesting to note that the Petrov Galerkin formulations, both on \((Q_2-Q_1)\) and \((Q_1-Q_1)\) spaces, as of their strong consistence could guarantee good stability versus accuracy compromise working with tuning parameters set to unit.

### 4 DOMAIN DECOMPOSITION METHOD

The implemented discretized DD method follows the explicit overlapping sub-domains approach (first proposed by Schwarz in the field of iterative solution of the classical elliptic boundary value problems). Such an approach is equivalent, from a performance viewpoint, to sub-structuring methods for advective-diffusive problems, as shown comparing the application of Schur methods (Robins-Robins or Neuman-Dirichlet like) with the Schwarz ones accelerated by Krylov subspace algorithm\(^2\). The adoption of DD entails the achievement of improved convergence rate and data locality for parallel application, and it allows the use of simple updating technique of the boundary data in each sub-domains. Although multiple level algorithm are more efficient (so called \textit{optimal algorithm}\(^2\)), relatively small numbers of sub-domains are here used, so that the application of single level method remains effective. Within the class of single level Schwarz methods (classical alternating, additive, and multiplicative) the chosen one is based on an additive algorithm in order to preserve the coding simplicity (with respect to the multiplicative one), and to guarantee reduced message passing costs (if compared to the alternating version).

<table>
<thead>
<tr>
<th>Schemes</th>
<th>Coefficients</th>
<th>(\frac{m_{\text{in}} - m_{\text{out}}}{m_{\text{in}}}) %</th>
<th>(\frac{P_{\text{in}}^{\text{av}} - P_{\text{out}}^{\text{av}}}{P_{\text{in}}^{\text{av}}}) %</th>
<th>(\frac{P_{\text{in}}^{\text{max}} - P_{\text{in}}^{\text{min}}}{P_{\text{in}}^{\text{min}}}) %</th>
</tr>
</thead>
<tbody>
<tr>
<td>(FV^{23})</td>
<td>(\varepsilon_2 = 0.3; \varepsilon_4 = 0.0; \varepsilon_{\text{pw}} = 0.0)(^7)</td>
<td>0.1</td>
<td>0.63</td>
<td>2.7</td>
</tr>
<tr>
<td>(FV^{23})</td>
<td>(\varepsilon_2 = 0.0; \varepsilon_4 = 0.25; \varepsilon_{\text{pw}} = 0.0)</td>
<td>0.11</td>
<td>-0.02</td>
<td>1.3</td>
</tr>
<tr>
<td>(FV^{23})</td>
<td>(\varepsilon_2 = 0.0; \varepsilon_4 = 0.25; \varepsilon_{\text{pw}} = 0.1)</td>
<td>-2.33</td>
<td>3.15</td>
<td>29.0</td>
</tr>
<tr>
<td>(SU-PS/PG, (Q1-Q1))</td>
<td>(\lambda_p = 0.3; \lambda_c = 1.0)</td>
<td>0.016</td>
<td>0.24</td>
<td>0.75</td>
</tr>
<tr>
<td>(SU-PS/PG, (Q1-Q1))</td>
<td>(\lambda_p = 0.3; \lambda_c = 0.05)</td>
<td>0.017</td>
<td>0.24</td>
<td>0.76</td>
</tr>
<tr>
<td>(SU/PG-PS/PG, (Q1-Q1))</td>
<td>(\lambda_p = 1.0; \lambda_c = 1.0)</td>
<td>0.017</td>
<td>0.0018</td>
<td>0.59</td>
</tr>
<tr>
<td>(SU/PG-PS/PG, (Q2-Q1))</td>
<td>(\lambda_p = 0.3; \lambda_c = 1.0)</td>
<td>0.009</td>
<td>0.087</td>
<td>0.68</td>
</tr>
<tr>
<td>(SU-PS/PG, (Q2-Q1))</td>
<td>(\lambda_p = 0.3; \lambda_c = 0.05)</td>
<td>0.009</td>
<td>0.087</td>
<td>0.68</td>
</tr>
<tr>
<td>(SU/PG-PS/PG, (Q2-Q1))</td>
<td>(\lambda_p = 1.0; \lambda_c = 1.0)</td>
<td>0.010</td>
<td>0.0012</td>
<td>0.38</td>
</tr>
</tbody>
</table>

\(^7\) \(\varepsilon_2\) and \(\varepsilon_4\) are respectively the coefficient for 2\textsuperscript{nd} and 4\textsuperscript{th} order artificial dissipation, \(\varepsilon_{\text{pw}}\) is the coefficient for pressure weighting\(^2\).

Table 2: Comparative analysis of stabilization effects among FV and FEM schemes
Concerning the implemented solution strategy, the developed DD approach, that authors have named *one level conforming inexact explicit non-linear overlapping Schwarz method*, updates at a time the conditions on the artificial sub-domain boundaries (that is of the whole inexact Oseen problem solution) and the non-linear advective and diffusive coefficients of system (19). Such original solution technique leads to accelerated convergence behavior\textsuperscript{26}. The independent sub-problems upon each sub-domains are solved using a block Gauss-Siedel preconditioned semi-iterative GMRes solver\textsuperscript{11,12,27}.

### 4.1 Variational interpretation of the Schwarz method

Let introduce the domain decomposition of discretized domain $\Omega$ into $n$ overlapping sub-domains such that $\Omega = \bigcup_i^\Omega_j$, denote the artificial boundaries by $\Gamma_{ij} = \partial U_i \cap \partial U_j$, $\forall i, j = 1, n$ with $j \neq i$, and the overlapping regions as $\Omega_{ij} = \Omega_i \cap \Omega_j$ (Figure 1).

The variational formulation of Schwarz additive method for the homogeneous Dirichlet boundary value problem (19) can be stated as follows:

set $V_i^h \in H_0^1(\Omega_i)$, for each domain $i = 1, n$ and for each step $k$

find $V_i^{h,k} \in H_0^1(\Omega_i)$ $\forall w_p^h \in V_{\partial \Omega_i}^h$, $\forall w_c^h \in V_{\partial \Omega_i}^h$, such that

\[
\begin{align*}
&c\left(\frac{-h,k}{i}, V_i^{h,k}, w^h\right) + \\
&s\left(\frac{V_i^{h,k}, w^h}{i}, \prod_i P_i \right) = \left(\phi B_i^h, w^h\right) + \left(\phi_{\partial \Omega_i}^h, w^h\right) - \left(\frac{F_i^k}{w^h}\right)_{\partial \Omega_i \cap \partial \Omega_j}^N \\
&U_i^{h,k+1} = U_i^{h,k} + \sum_i \tilde{V}_i^{h,k}
\end{align*}
\]

(21)

where $\tilde{V}_i^{h,k}$ identifies the extension of $V_i^{h,k}$ by 0 in $\Omega/\Omega_i$.

![Figure 1: Overlapping domain partition](image)

It is worth to note that in (21) the effect of Dirichlet boundary values on each subdomains solution $F_i^k\left(w^h\right)_{\partial \Omega_i \cup \partial \Omega_j, \Gamma_{ij}}$ account for both the domain condition applied to $\partial \Omega_D$ (depending from the original set of boundary values) and the artificial boundary values originating from
the neighboring subdomains $\Gamma_{ij}$ (function of the local inexact solution $V^{h,k}_j$). Due to the given definition of $\Gamma_{ij}$ two important features of artificial boundaries treatment algorithm should be pointed out\textsuperscript{26}. When $\Gamma_{ij}$ intersects $\partial\Omega_N$, the artificial Dirichlet boundary values is enforced instead of the original Neumann condition because of their explicit effect on the finite element equations. Then, in case of intersecting artificial boundaries ($\Gamma_{im} \cap \Gamma_{il} \neq \emptyset$), the Dirichlet artificial values applied to (21) are obtained as an average of local solutions computed independently for each node that falls into the multiple overlapping regions.

4.2 Parallel implementation

Among the different parallel methodologies a Single Program Multiple Data (SPMD) model is here applied. Following such an approach a two step procedure is adopted, based on both a DD and a message passing paradigm. The proposed DD approach has been developed in the framework of an *in-house* made code able to guarantee the load balancing among the processors and to bound the message-passing overhead. As mentioned above the computational domain is decomposed into a number ($n$) of sub-blocks, equal to the number of processors to be used, with a balanced sharing of computational load per processor. This operation on domain $\Omega$ could be schematized as follows

$$\Omega_i = \Lambda_i \Omega$$

(23)

where $\Lambda_i$ indicates the information transfer operator among the global domain nodes and the local subdomains ones (Figure 1). The overlap region is defined by a row of frame elements.

Since an explicit non-linear approach has been employed, the parallel solution of Navier-Stokes problem implies that each subdomain problem can be seen as independent and the coupling with the adjacent blocks is guaranteed by the overlapping regions. In such regions the value of variables arising from the neighboring subdomains are condensed and treated as Dirichlet pseudo-boundary conditions. For each explicit step a new set of pseudo-boundary condition is imposed. By using such a method the basic structure of the sequential solver is preserved since each processor operates on independent data-base except for the data communication required to update the pseudo-boundary conditions.

The communication phase requires to gather the pseudo-boundary values on a vector restricted to each subdomain. Then a communication structure with addressing/scattering arrays allows the correct distribution of the gathered data among processors. In the adopted communication strategy, managed with MPI libraries\textsuperscript{15}, each processor carries out successive *message-passing* iterations with the other ones. At the end of each step the synchronization among processors guarantees the correctness of the communication procedure, causing an overhead in the *message-passing* strategy. This notwithstanding the proposed procedure is able to minimize the dimension of the data exchange in each *send* or *receive* operation.
4.3 Computational characteristics of the implemented domain decomposition

The validation of the proposed parallel DD method developed within the FEM based Navier-Stokes solver is here carried out, in order to focus on the improvement of code computational performances after its parallelization as well on its capability of accurately predict the main phenomena arising in classical internal flow cases.

To evaluate the performance of implemented parallel computing algorithm two parameters have been considered. The speed-up factor defined as \( S(pr) = \frac{T(pr_{ref})}{T(pr)} \), where \( T(pr_{ref}) \) is the elapsed time spent to perform the computation using a reference number of processors \( (pr_{ref}) \), and \( T(pr) \) is the elapsed time spent for the calculation using \( (pr) \) processors. The efficiency is defined as \( E(pr) = \frac{S(pr)}{(pr/pr_{ref})} \). In the above the total time \( T(pr) \) is the sum of the time for serial code operations, the time required for the parallel computation, and the time spent during the communication phase. Concerning the FEM solution algorithm, all the simulations have been carried out using a finite element Petrov-Galerkin stabilized formulation with mixed element \( (Q2-Q1) \), and stabilizing coefficients are \( \lambda_p = 1.0; \lambda_c = 1.0. \) For the turbulent test case a standard EVM is used. The number of GMRes basis vectors is set equal to 50 to solve the inexact Oseen problem upon each subdomains. The convergence threshold for log of \( R_{res} \) and \( R_{sol} \) is set equal to -4.

4.3.1 Laminar flow in a 2D backward-facing step

In this test case, the flow field in a laminar backward-facing step is simulated and compared against the experiments of Armaly et al. The Reynolds number, based on the inlet section height \( (h) \) and the bulk velocity, is set equal to 300. Computations have been performed using a 81001 nodes grid (901 in the axial direction \( (x) \), 51 in the cross direction \( (y) \) upstream the step and 101 downstream). The 4 processors decomposition is based on crosswise artificial boundaries. While both 16 and 32 processors decompositions are obtained adding streamwise artificial boundaries. The calculations have been carried out using the Cray-T3E parallel machine. Table 3 shows the comparison between computed and measured distance of the reattachment point \( (x_s) \) from the edge of the step, as well as parallel performance parameters.

<table>
<thead>
<tr>
<th>Processors</th>
<th>Reattachment coordinate ( (x_s/h) )</th>
<th>( S(pr) )</th>
<th>( E(pr) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>6.32</td>
<td>4.00</td>
<td>1.00</td>
</tr>
<tr>
<td>16</td>
<td>6.32</td>
<td>15.38</td>
<td>0.96</td>
</tr>
<tr>
<td>32</td>
<td>6.32</td>
<td>21.63</td>
<td>0.68</td>
</tr>
<tr>
<td>Experiments</td>
<td>6.32</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 3: Accuracy evaluation and performance of parallel computations for 2D backward facing-step laminar flow

Data in Table 3 confirm that the re-circulation region is correctly predicted with all the
tested decomposition levels. As it could be expected, the efficiency of the code decreases with increasing processors number, in clear relation with the growth of convergence iterations and the communication times. Nevertheless $E(pr)$ values claim for good parallel performance.

### 4.3.2 Turbulent flow through a 2D 90° bend

The second flow case is used to show the efficacy of proposed DD method in simulating 2D turbulent flow through a 90° bend. The mean radius to duct width ratio was 2.3 and the straight inlet and outlet lengths were each 7.5 duct width long. The grid has 241 streamwise nodes 81 crosswise nodes, both uniformly distributed. The crosswise spacing sets $y^+$ in the range 30÷50. The decomposition uses different set of cross artificial boundaries. Uniform inlet profiles are specified for velocity, $k$ and $\varepsilon$, together with walls impermeability. The Reynolds number, based on the bulk velocity and the duct width, is $4 \times 10^4$. The calculation have been made using the SP2 IBM parallel machine. Table 4 shows the speed-up and efficiency factors for the set of tested artificial boundary configurations.

<table>
<thead>
<tr>
<th>Processors</th>
<th>$S(pr)$</th>
<th>$E(pr)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>2.00</td>
<td>1.00</td>
</tr>
<tr>
<td>3</td>
<td>3.16</td>
<td>1.05</td>
</tr>
<tr>
<td>6</td>
<td>6.28</td>
<td>1.05</td>
</tr>
</tbody>
</table>

Table 4: Performance of parallel computations for 2D turbulent flow in 90° bend

It is worth to note that an extra-convergence phenomenon appears increasing the subdomains number. Such an aspect could be related to the better convergence behaviour of local linearized problem in case of smaller subdomains.

### 4.3.3 Laminar flow in a 3D curved duct

The last validation test is the simulation of 3D laminar flow in a square duct 90° bend (40 mm of hydraulic diameter, 92 mm of mean radius and radius ratio 2.3), investigated by Taylor et al.\cite{Taylor1999}. The Reynolds number is equal 790 (based on bulk velocity $U_{ref}$ and hydraulic diameter $D_{ref}$). The results of the turbulent case are shown in Borello et al.\cite{Borello2000}.

![Figure 2: Velocity profiles in the middle of the cross section ($\lambda_R = 0.25$)\cite{Taylor1999}](image-url)
Due to the symmetry of the flow field, the computational domain models the lower part of the duct. The grid has 321 nodes in the streamwise (x) direction and 21×11 nodes in the cross (y,z) section uniformly distributed. The inlet velocity profile is derived from the experimental data set. The artificial surfaces fit to the channel cross sections. Computations have been made on the Cray-T3E parallel machine. Figure 2 shows the comparison between experimental and numerical velocity profiles along (z) direction, at the middle of channel cross section near to the duct exit with \( x_H = 0.25 \) (\( x_H \) is the dimensionless axial location downstream the bend). The analysis of the velocity profile confirms a fair agreement between computations and experiments, while Table 5 shows good parallel performance for 3D simulations.

<table>
<thead>
<tr>
<th>Processors</th>
<th>S(pr)</th>
<th>E(pr)</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>8.00</td>
<td>1.00</td>
</tr>
<tr>
<td>16</td>
<td>11.23</td>
<td>0.70</td>
</tr>
<tr>
<td>32</td>
<td>16.51</td>
<td>0.50</td>
</tr>
</tbody>
</table>

Table 5: Performance of parallel computations for 3D laminar flow in 90° bend

5 NUMERICAL ANALYSIS OF TURBOMACHINERY FLOWS

Two turbomachinery benchmark flows have been examined, both modelled under the incompressible flow hypothesis: a DCA cascade, a 3D isolated non-free vortex axial rotor.

5.1 Double Circular Arc cascade

This test case refers to the experimental work carried out by Zierke and Deutsch\(^{31}\) on a DCA compressor cascade. The numerical investigation has been limited to weak off-design condition with incidence angle equal to \(-1.5^\circ\), at a chord Reynolds number (based on the inlet flow velocity) of \(5.01\times10^5\). Such test case represents a severe challenge for the CFD codes, even at near-design conditions, due to the presence of large transition and separation phenomena arising along highly curved blade surfaces. The computational domain extends one chord upstream and downstream of the blade vane and is modelled with an H-type grid of 46737 nodes (577 streamwise and 81 pitchwise). Figure 3 shows grid details at the blade leading and trailing edges on the pressure surface. The inlet mesh angle is aligned with the incoming flow while the outlet one is placed equal to design outlet flow angle. The grid refinement is modulated to set the dimensionless distance \( y^+ \) value about 30 on the first nodes row near the solid boundaries. The adopted domain decomposition scheme consists of 6 subdomains specified by constant \( x \) artificial boundaries (see Figure 3). Non-linear EVM is used due to its ability of resolving turbulence anisotropy. As far as the boundary conditions are concerned, inlet uniform distributions are applied for velocity and turbulent variables. The inlet \( k \) value is obtained from the measured turbulence intensity \((TI = \sqrt{2k/3}/u_{inm} = 0.0018, \) where \( u_{inm} \) is the inlet velocity module), whereas the \( \varepsilon \) value is modelled on the basis of the characteristic length scale \((l^* = k^{1/2}/\varepsilon) \) assumed to be 0.004 of the chord\(^{32}\). Homogeneous
Neumann conditions are imposed on outlet section, and a synthetic wall treatment is employed on the blade profile. Flow periodicity is strictly imposed at the permeable boundaries. The simulation has been carried out using mixed order Petrov-Galerkin (Q2-Q1) stabilized elements, on a IBM SP2 machine.

Table 6 presents a comparison between measured and predicted pitchwise averaged cascade parameters: the diffusion factor $\hat{D}$, the global pressure coefficient $\hat{C}_{p\text{out}}$, the total pressure loss coefficient $\hat{\omega}$. The adopted parameters definitions follow

$$\hat{D} = \left( 1 - \frac{\hat{u}_{\text{out}}}{u_{\text{in}}} \right) + \frac{u_{\text{in}}^2 - u_{\text{out}}^2}{2\sigma u_{\text{in}}}$$

$$\hat{C}_{p\text{out}} = \frac{\hat{p}_{\text{out}} - \hat{p}_{\text{in}}}{0.5 \rho u_{\text{in}}^2}$$

$$\hat{\omega} = \frac{P_{\text{out}} - P_{\text{in}}}{0.5 \rho u_{\text{in}}^2}$$

where, the hat indicates blade-passage average, subscript $t$ refers to the tangential component, $u$ is the velocity norm, and $\sigma$ is the cascade solidity.

<table>
<thead>
<tr>
<th></th>
<th>$\hat{D}$</th>
<th>$\hat{C}_{p\text{out}}$</th>
<th>$\hat{\omega}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental data</td>
<td>0.55</td>
<td>0.47</td>
<td>0.09</td>
</tr>
<tr>
<td>Numerical data</td>
<td>0.58</td>
<td>0.56</td>
<td>0.07</td>
</tr>
</tbody>
</table>

Table 6: Global pitchwise averaged parameters, DCA cascade

Data in Table 6, confirm the agreement between the predicted flow behaviour and experiments. It is furthermore interesting to note that the slight numerical differences (respectively an over-prediction of diffusion factor and pressure coefficient and an under-
prediction of total loss coefficient) indicate that the boundary layer separation is not correctly predicted. The comparison of computed and measured pressure coefficient \( C_p = (p - p_{in})/(0.5 \rho u_{in}^2) \) distributions is shown in Figure 4, and puts forward the ability of the proposed code. The computed \( C_p \) distributions shows a fair agreement with the experiments along the pressure surface where the flow develops under an initial small favourable gradient region (up to 3\% chord), followed by a mildly adverse pressure gradient (3\%÷60\%). Downstream this location a favourable pressure gradient acts. From the experimental evidence transition occurs along that surface. It is worth noting that the prediction of the pressure distribution is good though the inability of the turbulence model to predict the transition effects. The analysis of \( C_p \) distributions along suction surface shows an initial adverse pressure gradient zone followed by a small region with favourable pressure gradient (3\%÷6\%). An adverse pressure gradient region then appears up to 80\% chord. Downstream the experimental pressure distribution shows a plateau, indicating a possible separation region near the trailing edge. The computational results agree well with the experiments except for the trailing edge region where the pressure gradient does not vanish, confirming that the adopted EVM is not be able to give a credible prediction of the separation phenomena.

Figure 5 shows the boundary layer profiles at three different chordwise locations along the suction side. In the first two locations the computational results are in fair qualitative agreement with experimental data, although numerical differences appear that could be related to the high-Re turbulence modelling. In the last location the presence of a large recirculating zone becomes evident in the measurements, that is absent in the prediction mainly because of the limit affecting the implemented EVM.

5.2 Non-free vortex axial fan rotor
The tested fan designed for non-free vortex (NFV) operation\textsuperscript{33,34}, consists of 12 straight (unswept) cambered plate blades, with a constant chord of 171 mm. The average chordwise tip clearance is 3 mm and the maximum blade thickness is 2 mm. The simulation has been carried out at design condition, with a rotational speed of 1100 rpm and Reynolds number (based on chord length and relative inlet flow speed at midspan) equal to $4 \times 10^5$. The domain defined by blade geometry and tip clearance, is modelled by a non-orthogonal body fitted coordinate system consisting of $81 \times 31 \times 41$ nodes in streamwise, blade to blade and spanwise direction respectively, appropriately stretched toward solid boundaries. Due to the thin blade profile an H-grid topology with the ‘pinching’ of blade tip grid points is used to model the clearance. The consequent high skewing of the mesh in that region is able to provide only a crude estimate of leakage flow phenomena\textsuperscript{35,36}. The adopted DD scheme generates 5 subdomains specified by artificial boundaries fitting to plane orthogonal to the axis. The inlet Dirichlet conditions for the relative velocity components are obtained from the LDA upstream measurements\textsuperscript{33}, while the inlet distribution of $k$ is obtained from axi-symmetric turbulence intensity profile derived on the basis of the upstream LDA data provided by Vad\textsuperscript{37}.

The inlet profile of $\varepsilon$ rate is based on the characteristic length scale definition (0.01 of rotor pitch at midspan). A standard EVM is used with a synthetic wall treatment able to simulate the effect of relative casing wall motion. Neumann homogeneous outflow conditions are specified for $k$ and $\varepsilon$ and non-homogeneous for the static pressure. The simulation has been carried out using (Q1-Q1) stabilized elements, on a IBM SP2 machine. The aim of the investigation is to focus on the capability of the proposed DD technique in predicting valid flow behaviour in realistic, decelerating and rotating cascades already studied using the serial FEM solver version\textsuperscript{36}. The ideal total head rise coefficient distributions are presented in Figure 6 (in case of zero inlet swirl defined as: $\psi_3 = 2R \left( u_3 p / U_c \right)$, with $R$ non-dimensional.

![Figure 5: Comparison of computed and measured velocity profile on suction surface](image)
radius, \( U_c \) peripheral velocity at the casing, \( u_{3p} \) local absolute peripheral velocity), where the measuring section is mapped onto its rectangular image. The predicted distribution shows a fair qualitative and quantitative agreement with the experiments in the wake (W) and the hub region (passage vortex PV and corner stall ST). Instead, a lack of agreement appears clearly in comparing the casing flow region, where the predicted behaviour highlights the decay of leakage flow influence (under-turning effect) at the aft part of the blade. Whereas the leakage flow effect appears strongly in LDA measurements, enforcing the NFV tip vortex structure and washing the annulus region. As far as the disagreement in the clearance region is concerned, the present Navier-Stokes solver uses a very crude representation of the blade tip, which is not able to solve the flow in the clearance space accounting for the blade thickness influence. Such inability is strengthened by the use of the synthetic treatment of casing wall, that fails in the prediction of physical velocity and force defect within the layer.

6 CONCLUSIONS

A parallel domain decomposition finite element formulation for modelling steady, 3D, incompressible and turbulent flow problems in turbomachinery applications was presented. Benchmark 2D and 3D steady fluid flows were simulated and both the accuracy and the features of the present method was demonstrated in the limits of the adopted turbulent modelling. The parallel solving procedure shows satisfactory speed-up and efficiency values for all the tested level of parallelization.

ACKNOWLEDGEMENTS

The authors would like to thank MURST (COFIN 1999), and CINECA under HPC Grant.
REFERENCES
