Computers & Fluids 110 (2015) 96–107

Contents lists available at ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

A parallel adaptive nonlinear elimination preconditioned inexact Newton method for transonic full potential equation

Feng-Nan Hwang [a,*], Yi-Cheng Su [a], Xiao-Chuan Cai [b]

[a] Department of Mathematics, National Central University, Jhongli 32001, Taiwan
[b] Department of Computer Science, University of Colorado, Boulder, CO 80309, USA

ARTICLE INFO

Article history:
Received 1 November 2013
Accepted 2 April 2014
Available online 18 April 2014

Keywords:
Transonic flow
Adaptive nonlinear elimination
Inexact Newton
Local high nonlinearity
Density upwinding finite difference
Shock wave

ABSTRACT

We propose and study a right-preconditioned inexact Newton method for the numerical solution of large sparse nonlinear system of equations. The target applications are nonlinear problems whose derivatives have some local discontinuities such that the traditional inexact Newton method suffers from slow or no convergence even with globalization techniques. The proposed adaptive nonlinear elimination preconditioned inexact Newton method consists of three major ingredients: a subspace correction, a global update, and an adaptive partitioning strategy. The key idea is to remove the local high nonlinearity before performing the global Newton update. The partition used to define the subspace nonlinear problem is chosen adaptively based on the information derived from the intermediate Newton solution. Some numerical experiments are presented to demonstrate the robustness and efficiency of the algorithm compared to the classical inexact Newton method. Some parallel performance results obtained on a cluster of PCs are reported.

1. Introduction

The class of inexact Newton method (IN) [8,15] is popular for solving large sparse nonlinear system of equations arising from discretization of partial differential equations (PDEs). IN is quite robust and efficient for smooth nonlinear problems, but if the solution of the problem or its derivatives has certain discontinuity, the convergence rate of IN degrades, and the method may fail to converge even used together with globalization techniques, such as linesearch or trust region [8,15]. Such problems appear often in computational fluid dynamics involving, for examples, shock waves, boundary layers, and corner singularities. To overcome the problem, we develop a nonlinear preconditioning technique in this paper.

Nonlinear preconditioning can be applied on the left or on the right of the nonlinear function. The basic idea of left preconditioning is to change the function of the system to a more balanced system and then solve the new system by IN. The additive Schwarz preconditioned inexact Newton algorithm (ASPIN) [3,11] belongs to this class. ASPIN has been applied successfully to incompressible high Reynolds number flows [3,4,6,11,12], transonic compressible flows [5,13,20], flows in porous media [19], unconstrained optimization problems arising in nonlinear elasticity problems [9], and image processing [22]. On the other hand, right preconditioning is to modify the variable of the nonlinear system. For example, Hwang et al. [13] employs a nonlinear elimination (NE) technique [14] as a right preconditioner for a quasi one-dimensional shocked duct flow calculation. The key idea of NE is to implicitly remove these components that cause trouble for IN. In order to use the algorithm proposed in [13], one has to assume that the components to be eliminated are known in advance. However, in practice it may not always be possible to determine these components. The main contribution of this paper is to propose a new algorithm, namely an adaptive nonlinear elimination (ANE) preconditioned inexact Newton method that does not require this assumption. In the proposed algorithm, we use the intermediate IN solution to identify these components to be eliminated before a new global IN iteration. One potential application of the proposed algorithm is for the time-dependent PDE problems solved by a fully implicit scheme. In this paper, we focus only on a steady-state problem, namely the full potential equation in two different computational domain. The subspace correction phase can be done before a global Newton iteration is performed so that the overall performance of IN-based kernel solver is improved.

The rest of the paper is organized as follows. The next section describes the full potential equation discretized using a finite difference method with density upwinding. Section 3 provides a detailed description of the proposed algorithm. Section 4 presents the numerical results, including parallel performance of the proposed algorithm. Section 5 summarizes the main contributions of

* Corresponding author. Tel.: +886 3 422 7151x65110; fax: +886 3 425 7379.
E-mail address: hwangf@math.ncu.edu.tw (F.-N. Hwang).

http://dx.doi.org/10.1016/j.compfluid.2014.04.005
0045-7930/

this paper and points out some potential applications of the algorithm.

2. Full potential flow equation and its discretization

We consider the full potential flow equation [10,18], which is often used for modeling transonic flows passing an airfoil,

$$\nabla \cdot (\rho(\phi)\nabla\phi) = 0, \tag{1}$$

where ϕ is the velocity potential, $(u_1, u_2) = \nabla\phi$ is the velocity field, and the density function ρ is given as

$$\rho(\phi) = \rho_\infty \left(1 + \frac{\gamma-1}{2} M_\infty^2 \left(1 - \frac{\|\nabla\phi\|_2^2}{q_\infty^2}\right)\right)^{1/(\gamma-1)}. \tag{2}$$

Here, $\gamma = 1.4$ is the specific heat for air. The constants ρ_∞, M_∞ and q_∞ represent the density, the Mach number, and the speed at the far field, respectively. In this work, two test cases, namely a flow passing the NACA0012 airfoil case [2,18], and a channel flow passing through a circular bump [7,16].

The geometrical configuration for a transonic flow passing the NACA0012 airfoil is shown in the left figure of Fig. 1, where the computational domain is $[0,1]\times[0,1]$ and the shape of the NACA0012 model is described by the function

$$f(x) = 0.17814(\sqrt{x} - x) + 0.10128(x(1-x)) - 0.10968x^2(1-x) + 0.06090x^3(1-x),$$

for $x \in (0,1)$ and then re-scaled into $[1/3, 2/3]$ through $x = 3t - 1$, for $t \in [1/3, 2/3]$. The boundary conditions are specified as follows.

1. $\phi = 0$ on the inflow boundary Γ_6, $\phi = q_\infty$ on the outflow boundary Γ_4, and the freestream boundary on Γ_5, which are described by $\phi = \phi_\infty = \int_x q_\infty\, dx$. The freestream speed q_∞ is normalized to be 1.
2. A homogenous Neumann boundary condition is imposed on Γ_1, and Γ_3, i.e., $\frac{\partial\phi}{\partial y} = 0$. This condition implies that the flow is symmetric with respect to the boundaries and no flow penetrate through the boundaries.
3. A transpiration boundary condition is given on Γ_2 by $\frac{\partial\phi}{\partial y} = -q_\infty f'(x)$.

As a second example, we consider a channel flow as shown in the right figure of Fig. 1. The computational domain is defined as $[-1.0, 4.0]\times[0.0, 2.073]$. The shape of the bump is described by the function.

$$f(x) = 4tx(1-x)$$

for $0 \leqslant x \leqslant 1, t = 0.042$. The settings of the boundary conditions are similar to the airfoil case, except that the freestream boundary condition on Γ_5 is replaced by a homogenous Neumann boundary condition as on Γ_1 and Γ_3 and $\phi = 0$ and $\phi = 1$ on Γ_6 and Γ_4, respectively.

To discretize (1) by a finite difference method with density upwinding [10], we begin by introducing a set of mesh points, $(x_i, y_j), 0 \leqslant i \leqslant n_x$ and $0 \leqslant j \leqslant n_y$ with the mesh size $h_x = l_x/n_x$ and $h_y = l_y/n_y$, where l_x and l_y are the lengths of the computational domain in the x- and y-directions, respectively. Let $\Phi = [\phi_{ij}]^T$ be the numerical approximations at mesh points (including the Dirichlet and Neumann boundary points) in the natural ordering. We denote $x_{i+1/2}$ and $y_{i+1/2}$, as the midpoints of subintervals $[x_i, x_{i+1}]$ and $[y_j, y_{j+1}]$, respectively. We discretize the full potential Eq. (1) at the interior point (x_i, y_j) using a second-order centered finite difference method, i.e.,

$$F_I(\Phi) \equiv h_y\left[\rho_{i+\frac{1}{2}j}(u_1)_{i+\frac{1}{2}j} - \rho_{i-\frac{1}{2}j}(u_1)_{i-\frac{1}{2}j}\right] + h_x\left[\rho_{ij+\frac{1}{2}}(u_2)_{ij+\frac{1}{2}} - \rho_{ij-\frac{1}{2}}(u_2)_{ij-\frac{1}{2}}\right] = 0,$$

where the velocity components u_1 and u_2 at $(x_{i+1/2}, y_j)$ and $(x_i, y_{j+1/2})$, respectively, are approximated as

$$(u_1)_{i+1/2,j} \approx (\phi_{i+1,j} - \phi_{ij})/h_x \tag{3}$$

$$(u_2)_{ij+1/2} \approx (\phi_{ij+1} - \phi_{ij})/h_y \tag{4}$$

and $(u_1)_{i-1/2,j}$ and $(u_2)_{ij-1/2}$ are approximated similarly. For purely subsonic flows, using (2) for calculating the flow density at $(x_{i\pm1/2}, y_j)$ and $(x_i, y_{j\pm1/2})$, i.e., $\rho_{i\pm1/2,j} = \rho(\|q\|_{i\pm1/2,j})$ and $\rho_{ij\pm1/2} = \rho(\|q\|_{ij\pm1/2})$, is sufficient. Here, $q_{i+1/2,j} = \sqrt{(u_1)_{i+1/2,j}^2 + (u_2)_{i+1/2,j}^2}$ and $q_{ij+1/2} = \sqrt{(u_1)_{ij+1/2}^2 + (u_2)_{ij+1/2}^2}$ with $(u_1)_{i+1/2,j}$ and $(u_2)_{ij+1/2}$ defined as (3) and (4), and

$$(u_2)_{i+1/2,j} \approx (\phi_{i+1,j+1} + \phi_{ij+1} - \phi_{i+1,j} - \phi_{ij})/(2h_y)$$
$$(u_1)_{ij+1/2} \approx (\phi_{i+1,j+1} + \phi_{i+1,j} - \phi_{ij+1} - \phi_{ij})/(2h_x).$$

However, for transonic flows, this formulation needs to be modified in order to capture the shock. By applying a first-order density upwinding scheme as suggested by Young et al. [2,21], a modified

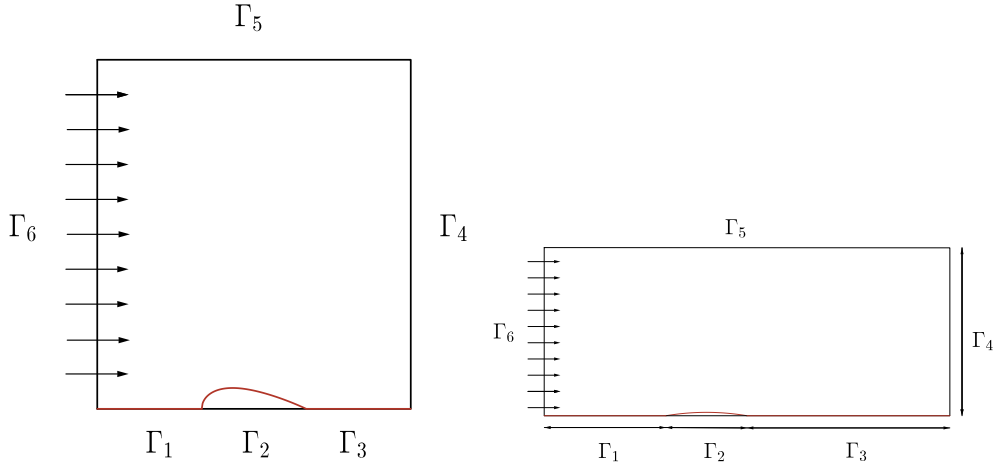

Fig. 1. The geometrical configuration for the airfoil problem (left) and the internal channel flow with a circular bump problem (right).