V European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2010
J. C. F. Pereira and A. Sequeira (Eds)
Lisbon, Portugal,14-17 June 2010

A HIGH-PERFORMANCE PARALLEL INCOMPRESSIBLE NAVIER-STOKES TWO-PHASE FLOW SOLVER USING THE LEVEL SET METHOD FOR HYDRODYNAMICS DESIGN

A.C. Lesage *, G. Houzeaux *, H. Coppola Owen *, M. Vazquez *

* Barcelona Supercomputing Center
Address : Edificio NEXUS I, planta 2 C/ Gran Capità, 2-4, 08034 Barcelona, Spain.
e-mail: {anne-cecile.lesage,guillaume.houzeaux,herbert.owen,mariano.vazquez}@bsc.es

Key words: Level Set Method, Finite Element Method,Variational multiscale method, parallel fractional predictor-corrector scheme, Orthomin iteration

Abstract. *Recent developments in robustness and accuracy of computational methods to predict flow of immiscible fluids separated by an interface made possible their application to hydrodynamics design. The main challenge of those methods is to predict: accurately, the global loads exerted by the water on marine structures (ships, offshore platforms, ..), and reliably local informations on the flow field making possible the optimization of the appendages and propulsive systems. Two important features of the computational method are required: a flexible unstructured discretization applied to locally-refined grids and high-performance parallelization of the CFD solvers able to run large 3D test cases (> 10 millions elements).*

In this paper, we propose the application of a massively parallel incompressible Navier-Stokes Eulerian two-phase flow solver to ship design process. The incompressible Navier-Stokes solver introduced in [8] is based on a parallel fractional predictor-corrector scheme implemented at the algebraic level, a Finite Element Method in space and implicit Finite Differences in time. It captures the interface motion on a fixed mesh using a Level Set method [24] discretized with the same features. The turbulence is modelled by the Spalart-Allmaras one-equation closure [27]. A Reichardt wall function permits to approximate the velocity field behavior in the boundary layer [20].

Our numerical method was tested against forces and wake shape experimental data on a benchmark widely used in the ship hydrodynamics community : the bare hull David Taylor Model Basin (DTMB) model 5512 at steady state [9, 12]. Our proposed scheme shows a very high scalability tested on a large number of processors (up to thousands).

1 INTRODUCTION

This paper studies the application to hydrodynamics of the fractional predictor-corrector solvers for incompressible flow introduced in [8] following a two-phase flow approach. Flows with interface motion (free surface and two-fluid interface problems) can be encountered in numerous fluid mechanics problems (ships, tsunamis, fluids in moving containers, mould filling). The methods used to simulate the evolution of the fluid interface can be classified in two categories: tracking techniques (moving mesh) and capturing techniques (fixed mesh).

In interface tracking techniques, the mesh is updated to track the interface by deformation for simple flows or by unsteady remeshing for complex flows [29, 30]. Although, such techniques give accurate interface representations, for complex 3D cases, they become too expensive. In interface capturing techniques, an interface function is used to capture the position of the interface within the resolution of a fixed computational mesh. This latter kind of techniques will be part of our proposal.

We propose a highly parallel numerical solver for the two-phase flow incompressible Navier-Stokes equations for hydrodynamics which consists of :

- a particular interface capturing technique, the Level Set method (LS),
- a time discretization based on implicit standard trapezoidal integration rule,
- a spatial discretization based on a stabilized Finite Element method referred to as algebraic sub-grid scales (ASGS),
- algebraic Richardson and Orthomin methods for the pressure Schur complement system that allow to obtain highly parallel solvers [7, 8],
- Reynolds Averaged Navier Stokes (RANS) equations using the Spalart-Allmaras (SA) one-equation closure for turbulence modelling [27],
- a Reichardt wall function for the viscous layer modelling [20].

The Level Set (LS) Method widely used for capturing interface evolution was introduced in the late 1980s by Osher and Sethian [15, 21]. It proposes to advect a relatively smooth field ϕ, the zero contour of which is the interface to represent. Since the function to advect is smooth (in contrast to the step function used by VOF methods), this method allows to obtain accuracy higher than first order. The LS method is appealing mainly for several reasons :

- it admits a convenient description of topologically complex interfaces, and it is quite simple to implement,
- it combines well with Finite Element discretization,

- it allows to discretize the problem on unstructured meshes,
- the interface location is accurately specified.

Conceptually, the ASGS method [3] is based on enlarging the finite element space by adding information about the part of the solution of the variational Galerkin problem that cannot be resolved by the computational grid. It permits to use equal interpolations for the velocity and the pressure (avoiding the need to satisfy the classical inf-sup condition) and deal with convection-dominated flows (cell Reynolds number greater than one).

Most of the fixed mesh methods for two-fluid phase flow share two basic steps. In the first step, an advection equation using the velocity field of the last time step is solved to allow to find the new interface position, and thus the material properties to be assigned in the next time step to solve the incompressible Navier-Stokes equation. In the second step, the two-phase flow Navier-Stokes equations are solved as one phase flow with variable properties.

A straightforward way to solve the discretized Navier-Stokes equations is to consider the monolithic scheme, that is to say to solve simultaneously the momentum and continuity equations in a coupled way. However, it is well known that, depending on the cases, iterative algebraic solvers (GMRES) could poorly converge for these coupled equations, unless a very robust preconditioner is used (like a so-called ILU preconditioner). Although sequentially efficient, these kind of preconditioners have bad speedup properties, precluding their use on large scale computers, where thousands of CPU's are involved in one single simulation because of its intensive communication needs.

One way of circumventing this fact is to "split" the discretized Navier-Stokes operator dividing the solution process in stages. From an algebraic point of view, this procedure consists in solving the pressure Schur complement system using an iterative method, via the solution of an intermediate velocity field. This technique enables to use adapted solvers to the momentum and Schur complement system equations. Traditional fractional steps techniques split the Navier-Stokes equations first, and then solve consecutively the momentum and continuity equations [2, 28, 17, 31, 19]. However, they introduce errors due the splitting. Instead, the predictor-corrector schemes based on Richardson and Orthomin methods considered here, converge to the monolithic solution. Thus they inherit the good algebraic properties given by the splitting of the momentum and continuity equations. The proposed schemes are both sequentially efficient and have shown a very high degree of scalability tested up to thousands of processors [7, 8].

The remainder of the paper is organized as follows. In section 2, we recall the main principles of the LS method and presents a particular implementation of this method with a finite element implicit discretization. In section 3, we describe the mathematical model used to solve the two-phase flow incompressible Navier-Stokes equations. In section 4, we present the main features of the parallel fractional predictor corrector schemes. In section 5, we adress the Spalart-Allmaras turbulence modelling. In section 6, we describe the viscous boundary layer treatment with the Reichardt wall function. Section 7, is devoted

to validation. We validate our Incompressible Navier-Stokes two-phase flow solver on the bare hull David Taylor Model Basin (DTMB) model 5512 at steady state [9, 12]. Two algorithm features are evaluated : its accuracy compared to experimental data and its parallel performance. Finally, some conclusions are given in section 8.

2 IMPLEMENTATION OF THE LEVEL SET METHOD

2.1 Level Set Principle

We assume that the interface divides the whole domain Ω in two disconnected domains so that $\Omega = \Omega_1 \cup \Omega_2$. The interface advection with a divergence free velocity field $\mathbf{U}$, can be written with the characteristic function χ so that:

$$\frac{\partial \chi}{\partial t} + \mathbf{U} \cdot \nabla \chi = 0, \text{ with } \chi(\mathbf{x}, t) = \begin{cases} = 1 & \text{if } \mathbf{x} \in \Omega_1 \\ = 0 & \text{if } \mathbf{x} \in \Omega_2 \end{cases} \tag{1}$$

The formal accuracy of the advection of a step function as χ is severely limited to first order. To tackle this problem, we can distinguish two main approaches in the literature :

- a numerical scheme that exploits the fact that χ takes only two different values. A particular way of doing this is the second-order VOF (see, for example [18]),
- χ is advected through an auxiliary smooth function. The LS method introduced by Osher and Sethian [15, 21] relies on a such a function ϕ satisfying $\chi = H(\phi)$.

In this paper, we choose the second option using the LS method. The underlying idea behind this method is to embed an interface Γ in $\mathcal{R}^3$, which bounds an open region $\Omega \in \mathcal{R}^3$ as the zero level of a higher dimensional function $\phi(\mathbf{x}, t)$. Let H be the step function $H(x) = 1$ if $x > 0$ and $H(x) = 0$ elsewhere.

$$\frac{\partial \phi}{\partial t} + \mathbf{U} \cdot \nabla \phi = 0 \tag{2}$$

The Level Set function is defined as a continuous function, initialized and periodically reset as the signed-distance to the interface $\phi = d(\Gamma)$ ($|\nabla \phi| = 1$):

$$\phi(\mathbf{x}, t) = \begin{cases} > 0 & \text{if } \mathbf{x} \in \Omega_1 \\ < 0 & \text{if } \mathbf{x} \in \Omega_2 \end{cases} \tag{3}$$

The interface Γ corresponds to the zero level (*Level Set*) of the function ϕ.

$$\Gamma = \{\mathbf{x} \mid \phi(\mathbf{x}, t) = 0\} \tag{4}$$

The motion of the interface is determined by the velocity field, $\mathbf{U}$, which for our case is given externally by the material velocity of the fluid flow simulation. Note that in principle the equation (2) only needs to be solved locally near the interface, cf. [1, 16].

2.2 Finite Element Implicit Discretization

Due to the pure convective type of equation (2), we use the SUPG technique to stabilize it. The time discretization is carried out using the generalized trapezoidal rule, i.e. a finite difference scheme. Let us introduce a uniform partition of the time interval $[0;T]$ and define

$$\begin{aligned} \phi^{n+\theta} &:= \theta\phi^{n+1} + (1-\theta)\phi^n && (5)\\ \delta t &:= t^n - t^{n-1} && (6)\\ \delta_t\phi^{n+\theta} &:= \frac{\phi^{n+\theta} - \phi^n}{\theta\delta t} && (7) \end{aligned}$$

where δt is the time step size and superscript n denotes the approximated solution at time $n\delta t$. The parameter $\theta \in [0,1]$ determines the order of the scheme. A first order scheme is obtained by choosing $\theta = 1$ (Euler) and a second order method is obtained with $\theta = 0.5$ (Crank–Nicolson). Let us define $\int_{\Omega'}(\cdot)d\Omega = \sum_K \int_{\Omega_K}(\cdot)d\Omega$ where Ω_K is the interior of element K of the partition. Then the discrete problem, both in space and time, is stabilized by the SUPG technique which is described as follows : Given a velocity at time $\mathbf{u}_h^{n+\theta}$ at time $t^{n+\theta}$ and a ϕ_h^n at time t^n, find $\phi_h^{n+\theta} \in V_h$ by solving the discrete variational problem :

$$\begin{aligned} & \int_{\Omega'} \ (\ \alpha\phi_h^{n+\theta} + \mathbf{u}_h^{n+\theta}\cdot\nabla\phi_h^{n+\theta} \ )(\ v_h + \tau\mathbf{u}_h^{n+\theta}\cdot\nabla v_h \ )d\Omega \\ = & \int_{\Omega'} \ (\ \alpha\phi_h^{n} \ )(\ v_h + \tau\mathbf{u}_h^{n+\theta}\cdot\nabla v_h \ )d\Omega \end{aligned} \quad (8)$$

with $\alpha = \frac{1}{\theta\delta t}$. The parameter τ is chosen in order to obtain a stable numerical scheme with optimal convergence rate. It is computed within each element domain Ω_e as :

$$\tau = \frac{h^e}{2|\mathbf{u}^e|} \quad (9)$$

where h^e is the element length in the direction of the flow and $|\mathbf{u}^e|$ the velocity norm of element e. Finally the LS at time step $n+1$ is computed as :

$$\phi_h^{n+1} = \phi_h^n + \frac{\phi_h^{n+\theta} - \phi_h^n}{\theta}. \quad (10)$$

2.3 Redistanciation

Initially $|\nabla\phi| = 1$, but when the Level Set function ϕ is advected, $|\nabla\phi|$ can tend to zero (for example in a case where the velocity field is stretching the LS iso-values). Then, the accuracy of the interface advection and the computation of interface properties (curvature and contact angle) is no more ensured cf [11]. This is one of the motivations of the periodical replacement of the ϕ solution of the advection problem by the signed distance to its zero level.

$$d(\phi)(\mathbf{x},t) = \text{sign}(\phi)\text{dist}(\mathbf{x},\Gamma(\phi)). \quad (11)$$

The replacement of ϕ by $d(\phi)$ is ensured at the discrete level by algorithms called redistanciation or reinitialisation. They should maintain as accurately as possible the level $\phi = 0$, which defines the interface and adjust the degrees of freedom of function ϕ closer to a signed distance to the interface.

In the literature, we can distinguish two types of redistanciation: by geometrical computation or by solving an equation cf. [21, 15, 25, 26].

For the redistanciation using an equation, we use the one proposed by Sussman *et al.* [25]:

$$d_\tau + \mathbf{w}.\nabla d \;=\; \text{sign}(\phi) \tag{12}$$

with $\mathbf{w} = \text{sign}(\phi)\frac{\nabla\phi}{|\nabla\phi|}$. The increasing complexity of the geometrical redistanciation makes it unfeasible for large 3D test cases. Thus in this paper, we use the Sussman one's. Note that we use the same SUPG stabilization for the redistanciation equation (12) as the one described for the LS advection (cf subsection 2.2) with $\mathbf{u}^{n+\theta} = \texttt{sign}(\phi)\frac{\nabla\phi}{|\nabla\phi|}$ and an additional right hand side term. But we solve it explicitly with first order in time.

2.4 Enforcing Mass Conservation

If no phase change occurs, the mass of each of the two fluids should be conserved. Being an incompressible flow, the volume occupied by any fluid should be preserved as well. Eventually, the LS method will introduce loss or gain of mass, the reasons are manifold: loss of steepness in the interface region, inexact divergence of the velocity field, error in advection, etc. Inacuracies accumulate in time. After several simulation time steps, the volume can vary widely. This lack of exact conservation of liquid mass has been reported repeatedly in the literature [26, 22, 14]. We can basically classify the methods to enforce mass conservation in two groups :

- The methods which consist in adding and removing mass in the interface region in order to obtain an exact conservation of mass. At the end of every timestep, the total amount of fluid mass is compared to the expected value. The expected value is determined from the mass at the previous timestep, plus the massflux across all boundaries during the timestep. The difference between the expected and actual mass is typically very small, then a quick convergence is achieved by simply adding and removing mass appropriately through an iterative Newton method. There are two versions of this correction : small uniform correction on all the interface [22] (c_ϕ= cst) and a correction proportional to the absolute value of the normal velocity of the interface [13] (c_ϕ= cst$|\mathbf{v}.\frac{\nabla\phi}{|\nabla\phi|}|$) .
- The methods which consist in improving the mass conservation in the advection step [14, 23].

In this paper, we use the uniform mass correction.

3 STABILIZED FINITE ELEMENT DISCRETIZATION OF THE TWO PHASE TURBULENT INCOMPRESSIBLE NAVIER-STOKES EQUATIONS

The velocity $\mathbf{u}$ and pressure p fields of two incompressible fluids moving in the domain $\Omega = \Omega_1 \cup \Omega_1$ during the time interval $(t_0,\, t_f\,)$ can be described by the incompressible two fluid RANS equations [26, 25]:

$$\begin{aligned} \rho[\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}.\nabla)\mathbf{u}] - \nabla.[2(\mu+\mu_t)\epsilon(\mathbf{u})] + \nabla p &= \rho\mathbf{f} \; in \; \Omega, \\ \nabla.\mathbf{u} &= 0 \; in \; \Omega \end{aligned}$$

where ρ is the density, μ the dynamic viscosity, μ_t the eddy dynamic viscosity (turbulence modelling), ϵ is the velocity strain rate where $\epsilon(\mathbf{u}) = \frac{1}{2}(\nabla\mathbf{u} + \mathbf{u}^t)$ and f the vector external body forces, which includes the gravity force $\rho\mathbf{g}$. The density, velocity, dynamic viscosity and pressure are defined as

$$\mathbf{u}, p, \rho, \mu = \begin{cases} \mathbf{u}_1, p_1, \rho_1, \mu_1 & \mathbf{x} \in \Omega_1 \\ \mathbf{u}_2, p_2, \rho_2, \mu_2 & \mathbf{x} \in \Omega_2 \end{cases}$$

where Ω_1 indicates the part of Ω occupied by fluid number 1 and Ω_2 indicates the part of Ω occupied by fluid number 2. The extent of Ω_1 and Ω_2 is given by the level set function ϕ (cf paragraph 2.1) together with initial and boundary conditions. To ensure stability the density and viscosity jumps are smoothed at the interface [26].

ASGS stabilization [3] is used to deal with convection-dominated flows and to circumvent the well known div-stability restriction for the velocity and pressure Finite Element space, allowing in particular equal interpolation for both unknowns.

Let $\mathbf{v}$ and q be the velocity and pressure test functions, respectively (belonging to V and Q which are the usual Sobolev spaces). The weak form of Navier Stokes equations writes:

$$\begin{aligned} (\rho\partial_t\mathbf{u}, \mathbf{v}) + a(\mathbf{u}, \mathbf{v}) - b(p, \mathbf{v}) &= (\rho\mathbf{f}, \mathbf{v}), \\ b(q, \mathbf{u}) &= 0 \end{aligned}$$

with $a(\mathbf{u}, \mathbf{v}) = (\rho(\mathbf{u}^i.\nabla)\mathbf{u}^i, v) + (2(\mu+\mu_t)\epsilon(\mathbf{u}),\ \epsilon(\mathbf{v}))$ and $b(q, \mathbf{v}) = \int_\Omega q\nabla.\mathbf{v}d\Omega$. For the linearization of the $\mathbf{u}^i.\nabla\mathbf{u}^i$ term, we use the Picard method. See [12] for an introduction and [13] for a complete mathematical analysis.

The space discretization using the stabilized Finite Element method with ASGS writes for the momentum equation :

$$\begin{aligned} & (\rho\partial_t\mathbf{u}_h, \mathbf{v}_h) + a(\mathbf{u}_h, \mathbf{v}_h) - b(p_h, \mathbf{v}_h) + s_1(\mathbf{u}_h, \mathbf{v}_h) + s_2(p_h, \mathbf{v}_h) \\ = \; & (\rho\mathbf{f}, \mathbf{v}_h) + (\mathbf{f}_\mathbf{u}, \mathbf{v}_h) \end{aligned} \tag{13}$$

with

$$
\begin{aligned}
s_1(\mathbf{u}_h, \mathbf{v}_h) &= \int_{\Omega'} \tau_1(\rho(\mathbf{u}_h^i \nabla)\mathbf{v}_h + \nabla.[2(\mu+\mu_t)\epsilon(\mathbf{v}_h)]).(\rho(\mathbf{u}_h^i.\nabla)\mathbf{u}_h - \nabla.[2(\mu+\mu_t)\epsilon(\mathbf{u}_h)])d\Omega \\
&+ \int_{\Omega'} \tau_2(\nabla.\mathbf{u}_h)(\nabla.\mathbf{v}_h)d\Omega, \\
s_2(p_h, \mathbf{v}_h) &= \int_{\Omega'} \tau_1(\rho(\mathbf{u}_h^i.\nabla)\mathbf{v}_h + \nabla.[2(\mu+\mu_t)\epsilon(\mathbf{v}_h)]).\nabla p_h \; d\Omega \\
(\mathbf{f}_\mathbf{u}, \mathbf{v}_h) &= \int_{\Omega'} \tau_1(\rho(\mathbf{u}_h^i.\nabla)\mathbf{v}_h + \nabla.[2(\mu+\mu_t)\epsilon(\mathbf{v}_h)]).\rho\mathbf{f} \; d\Omega
\end{aligned}
$$

and for the continuity equation:

$$
b(q_h, \mathbf{u}_h) + s_3(\mathbf{u}_h, q_h) + s_4(p_h, q_h) = (f_p, q_h) \tag{14}
$$

$$
\begin{aligned}
s_3(\mathbf{u}_h, q_h) &= \int_{\Omega'} \tau_1 \nabla q_h.(\rho(\mathbf{u}_h^i.\nabla)\mathbf{u}_h - \nabla.[2\mu\epsilon(\mathbf{u}_h)])d\Omega, \\
s_4(p_h, q_h) &= \int_{\Omega'} \tau_1 \nabla q_h.\nabla p_h d\Omega, \\
(f_p, q_h) &= \int_{\Omega'} \tau_1 \nabla q_h.\rho\mathbf{f} d\Omega
\end{aligned}
$$

The terms s_1, s_2, s_3, s_4, $(\mathbf{f}_\mathbf{u}, \mathbf{v}_h)$ and (f_p, q_h) are additional terms introduced by the ASGS stabilization mixing the momentum and continuity equations. See [6] for all the details of the formulation. Parameters τ_1 and τ_2 are usually referred to as the stabilization parameters, and they are computed elementwise. For this, we make use of [3]'s formulas

$$
\begin{aligned}
\tau_1 &= \left[\frac{4\mu}{h_e^2} + \frac{2\rho|\mathbf{u}_e|}{h}\right]^{-1}, \\
\tau_2 &= \frac{h_e^2}{\tau_1} = 4\mu + 2\rho|\mathbf{u}_e|h_e
\end{aligned}
$$

where h_e is the characteristic minimum element size and half of it for quadratic elements [3]. Note that the time discretization is carried out using the generalized trapezoidal rule as described in section 2.2 for the LS advection.

The set of discretized incompressible Navier-Stokes equations (13) and (14) can be reformulated in a matrix form as the following :

$$
\begin{bmatrix} \mathbf{A}_{uu} & \mathbf{A}_{up} \\ \mathbf{A}_{pu} & \mathbf{A}_{pp} \end{bmatrix} \begin{bmatrix} \mathbf{u} \\ \mathbf{p} \end{bmatrix} = \begin{bmatrix} \mathbf{b}_u \\ \mathbf{b}_p \end{bmatrix}
$$

The solution of this system, using a direct solver or an iterative solver with preconditioning is referred as monolithic scheme.

4 PARALLEL PREDICTOR CORRECTOR SCHEMES USING PRESSURE SCHUR COMPLEMENT

The parallel predictor corrector schemes we use are based on the solution of the pressure Schur complement system [5]. This consits of the pressure equation obtained after eliminating the velocity from the momentum equation by manipulating the previous matrix system. It reads as follows :

$$\mathbf{S}\mathbf{p} = \mathbf{b}_s \tag{15}$$

with

$$\begin{aligned} \mathbf{S} &= \mathbf{A}_{pp} - \mathbf{A}_{pu}\mathbf{A}_{uu}^{-1}\mathbf{A}_{up} \\ \mathbf{b}_s &= \mathbf{b}_p - \mathbf{A}_{pu}\mathbf{A}_{uu}^{-1}\mathbf{b}_u \end{aligned} \tag{16}$$

Assembling the previous system is obviously not viable since it involves the inverse $\mathbf{A}_{uu}^{-1}$. The predictor-corrector schemes we use are based on solving iteratively for the Schur complement with appropriate preconditioners $\mathbf{Q}$ [8]. The proposed schemes are based on four main blocks.

- Momentum block
  1. Solve Momentum equation $\mathbf{A}_{uu}\mathbf{u}^{k+1} = \mathbf{b}_u - \mathbf{A}_{up}\mathbf{p}_k$
  2. Update Schur complement system $\mathbf{r}^k = \mathbf{b}_s - \mathbf{S}\mathbf{p}^k$
- Richardson block
  1. Solve Schur complement equation $\mathbf{Q}\mathbf{z} = \mathbf{r}^k$
  2. Update pressure $\mathbf{p}^{k+1} = \mathbf{p}^k + \mathbf{z}$
- Orthomin block
  1. Solve Schur complement equation $\mathbf{Q}\mathbf{z} = \mathbf{r}^k$
  2. Solve Momentum equation $\mathbf{A}_{uu}\mathbf{v} = \mathbf{A}_{up}\mathbf{z}$
  3. Compute $\mathbf{x} = \mathbf{A}_{pp}\mathbf{z} - \mathbf{A}_{pu}\mathbf{v}$
  4. Compute $\alpha = \frac{\langle \mathbf{r}^k, \mathbf{x} \rangle}{\langle \mathbf{x}, \mathbf{x} \rangle}$
  5. Update velocity $\mathbf{u}^{k+1} = \mathbf{u}^k - \alpha\mathbf{v}$
  6. Update pressure $\mathbf{p}^{k+1} = \mathbf{p}^k - \alpha\mathbf{v}$
  7. Update Schur complement residual $\mathbf{r}^{k+1} = \mathbf{r}^k - \alpha\mathbf{x}$
- Correction block

1. Update velocity $\mathbf{u}^{k+1} = \mathbf{u}^k + \mathbf{C}(\mathbf{p}^{k+1} - \mathbf{p}^k)$

In [8] we introduced two preconditioners. The first one is the well-known $\delta t \Delta$ operator used in classical fractional step techniques. The second one is based on the approximate weak form of Uzawa operator $\nabla.(\tau_1 \nabla)$, where τ_1 is the classical stabilization parameter, that is the local inverse of the momentum operator used in variational multiscale methods [7].
$\mathbf{C}$ is a correction matrix such that the continuity equation is almost satisfied at iteration $k+1$ up to an error vector ε such that

$$\mathbf{A}_{pp}\mathbf{p}^{k+1} + \mathbf{A}_{pu}\mathbf{u}^{k+1} = \mathbf{b}_p + \varepsilon \tag{17}$$

In the Orthomin iteration α is a relaxation parameter computed dynamically in order to minimize the norm of the residual at next iteration $||r^{k+1}||^2$.

Richardson and Orthomin iterations are defined according to the previous blocks (cf [8] for more details):

- Momentum preserving Richardson iteration (MR)
  1. Call Momentum block
  2. Call Richardson block
  3. Call Momentum block
- Continuity preserving Richardson iteration (CR)
  1. Call Momentum block
  2. Call Richardson block
  3. Call Correction block
- Momentum preserving Orthomin iteration (MO)
  1. Call Momentum block
  2. Call Orthomin block
- Continuity preserving Orthomin iteration (CO)
  1. Call Momentum block
  2. Call Orthomin block
  3. Call Richardson block
  4. Call Correction block

The Continuity preserving Richardson iteration corresponds to the traditional fractional scheme [2, 28]. As we seek for transient solutions, we use the previous algorithms as predictor-corrector like schemes. This is achieved by coupling the linearization and Schur complement solver loops (index noted as i). In this case, and if convergence is achieved, the solution at each time step is the same as that of the monolithic scheme.

5 SPALART-ALLMARAS TURBULENCE MODELLING

The turbulence model chosen to compute the eddy kinematic viscosity $\nu_t = \mu_t/\rho$ is the Spalart-Allmaras turbulence model. This model was devised "using empiricism and arguments of dimensional analysis, Galilean invariance, and selective dependence on molecular viscosity" [27]. It consists of a transport equation for the eddy viscosity ν_t. For any details on the equation, see the original publication of the authors [27]. The equation writes as :

$$\partial_t \nu_t + \mathbf{u}.\nabla\nu_t - c_{b_1} S\nu_t - \frac{1}{\sigma}[\nabla.(\nu_t\nabla\nu_t) + c_{b_2}(\nabla\nu_t)^2] + c_{w_1} f_w \frac{\nu_t^2}{d^2} = 0 \text{ in } \Omega \times (0,T), \quad (18)$$

where c_{b_1}, c_{b_2}, σ and c_{w_1} are constants, S is the norm of the vorticity, f_w is a function depending on S, ν_t and the distance to the wall d. This equation is the high Reynolds number version of the model. Additional corrections enable for to compute low Reynolds number and transition effects. This equation must be supplied with appropriate and boundary conditions.

6 REICHARDT WALL FUNCTION APPROACH

The RANS and turbulence equations can be solved using the wall function approach on the boundary of the computational domain. The wall function approach implemented in this work consists in assuming that the computational wall Γ_M is located sufficiently far from the real wall (variable on Γ_M are identified with a hat) where the no-slip condition for the velocity holds. It avoids solving for the large gradients present in the boundary layer. Then we use a model equation for the tangential traction $\mathbf{t}_t$ together with a non-penetrating condition for the velocity $\hat{\mathbf{u}}$ as well as a model equation for the eddy viscosity (cf Figure 1):

$$\mathbf{t}_t = -\frac{\rho U_*^2 \hat{\mathbf{u}}}{|\hat{\mathbf{u}}|} \quad (19)$$

where U_* is the friction velocity. We define $\hat{U}^+ = \frac{|\hat{\mathbf{u}}|}{U_*}$, $\hat{y}$ the prescribed normal distance of Γ_M to the wall and $\hat{y}^+ = \frac{\hat{y}U_*}{\nu}$ the non-dimensional distance to the wall. The so-called Reichardt's law which writes as follows :

$$\hat{U}^+ = \frac{1}{\kappa}\ln(1+0.4\hat{y}^+) + 7.8[1-\exp(\frac{\hat{y}^+}{11}) - \frac{\hat{y}^+}{11}\exp(-0.33)\hat{y}^+], \quad (20)$$

enables one to estimate U_* from the value of $\hat{\mathbf{u}}$ known from a previous iteration (using a Newton-Raphson scheme).

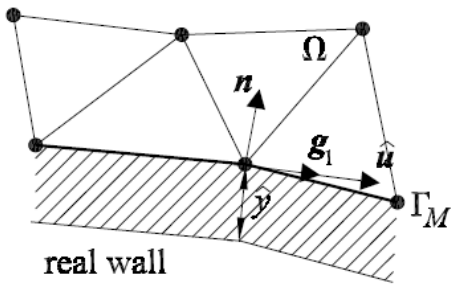

Figure 1: Local system on the boundary Γ_M

The wall condition for the eddy viscosity is computed using the classical mixing length hypothesis together with the Van-Driest damping function, i.e., we impose that

$$\nu_t = l_{\text{mix}} |\partial u / \partial y|, \tag{21}$$

where the mixing length is given by $l_{\text{mix}} = \kappa \hat{y}^+ [1 - \exp(\hat{y}^+ - /26)]$.

7 NUMERICAL VALIDATION

7.0.1 Kelvin wake around steady DTMB 5512 model

The problem under study is the computation of the steady Kelvin wake around a US Navy surface combatant restrained from motions, advancing in flat sea at constant speed ($Re = 4.78.10^6$ (water), $Fr = 0.28$). The ship is the bare hull David Taylor Model Basin (DTMB) model 5512, a 1:46.6 model scale of a modern surface combatant. The geometry is a benchmark on the ship hydrodynamics community, tested in the towing tanks at DTMB, IIHR (Iowa) [9] and INSEAN (Italy) [10]. It has a sonar dome, which provides additional geometric complexity. The DTMB 5512 model is L = 3.048 m long with 0.132 m draft.

The computational domain extends from -L $\leq$ x $\leq$ 2 L, -L $\leq$ y $\leq$ L, -0.2 L $\leq$ z $\leq$ 0.2 L for the medium speed case in dimensionless coordinates. The simulation is performed for the entire ship-hull. The ship axis is aligned with the x-axis with the bow at x = 0 and the stern at x = 1. The free surface at rest lies at z = 0. At t = 0, the ship is accelerated impulsively to full speed. Figure 2 shows a cut at $y = 0$ of the tetrahedra mesh. The mesh is refined in the vicinity of the whole interface and the ship geometry. It is not refined in the boundary layer.

Figure 3 shows the comparison of the convergence on the momentum and continuity residuals for four predictor-corrector scheme options : Momentum preserving Richardson iteration (MR), Continuity preserving Richardson iteration (CR), Momentum preserving Orthomin iteration (MO) and Continuity preserving Orthomin iteration (CO) (cf section 3.1). The MR options diverges on this benchmark, the CR options does not show to be

robust either. Both options with Orthomin iteration show good convergence. The MO option gives better convergence on the momentum residual and the CO on the continuity one, as expected. For the DTMB 5512 steady state simulation, we choose to use the MO iteration which is less demanding in computation terms. Table 1 gives a sum up of the parameters of the numerical simulation.

Figure 2: Cut at $y = 0$ of the DTMB5512 tetrahedra mesh

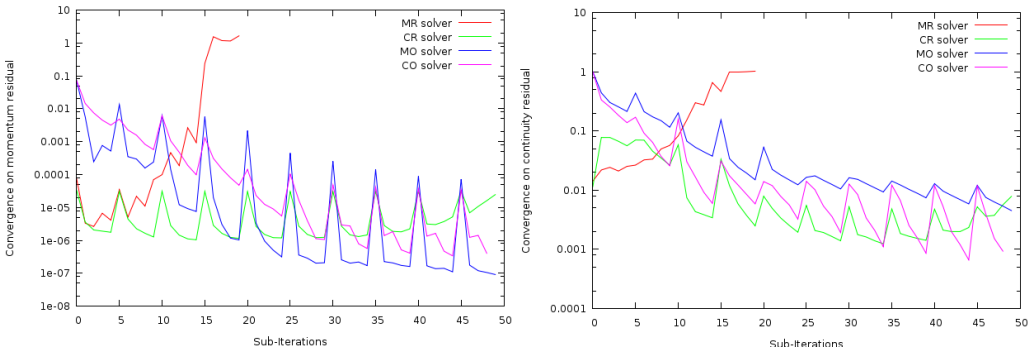

Figure 3: Convergence comparison of four Predictor Corrector Scur complement solver scheme options (MR, CR, MO, CO) on Momentum (Left) and Continuity (Right) residual convergence

The steady Kelvin wake establishes after a simulation time of $12s$ (cf Figure 4). Figure 5 shows a good agreement for the established wave elevation profile between the numerical results and for the experimental data [12].

Table 2 compares the experimental values and the numerical values for some forces exerted by the water on the ship. The evaluated forces are the drag force F_x, the viscous

Mesh size	:	2526423 nodes, 14553898 elements (tetrahedra).
Domain size	:	3L × 2L × 0.4L (3D unstructured mesh)
Interface height at rest	:	$a = 0.0m$
Density	:	$\rho_l = 997.8\ kg/m^3\ \rho_g = 1.2\ kg/m^3$
Viscosity	:	$\mu_l = 9.772.10^{-4}\ kg/m\ s,\ \mu_g = 1.7.10^{-5}\ kg/m\ s$
Volumic forces (gravity)	:	$g = -9.81\ m.s^{-2}$
Entrance and exit boundary conditions	:	prescribed velocity $U = 1.536\ m.s$
Boundary conditions	:	Reichardt wall function, $\hat{y} = 0.006$
Turbulence modelling	:	RANS using SA
Time step	:	$\Delta t = 0.01s$
Predictor corrector scheme	:	Momentum preserving Orthomin iteration (MO)
Volume correction	:	uniform
Redistanciation	:	solving equation (12)
Momentum Solver	:	BI CGSTAB
Continuity Solver	:	Deflated CG

Table 1: Parameters for the numerical simulation of the steady DTMB 5512 model

force F_{vx}, the pressure force F_{px}, the drag coefficient $C_T = \frac{F_x}{0.5\rho U_c^2 S}$ and the frictional drag coefficient $C_F = \frac{F_{vx}}{0.5\rho U_c^2 S}$ [9]. $S = 1,370\ m^2$ is the static wetted surface. C_F can evaluated using the model-ship correlation line (ITTC 1957) $C_F = \frac{0.075}{(log_{10}Re - 2.0)^2} = 3.425\ 10^{-3}$.

Value	Experimental or formula value	Numerical value new mesh	Relative Error
F_x	7.432 N	7.28 N	2%
F_{vx}	5.52 N	5.24 N	4.71%
F_{px}	1.912 N	2.04 N	6.7 %
C_T	4.607 10^{-3}	4.514 10^{-3}	
C_F	3.425 10^{-3}	3.249 10^{-3}	

Table 2: Comparison of experimental and numerical data on drag forces and coefficients. at $t = 12s$.

The relative error between the experimental value and the numerical value for C_T is in the range of uncertainty analysis described in [12]. Thus our simulation gives a relatively accurate prediction of the forces exerted on the boat.

To measure the parallel performance of our Predictor Corrector Incompressible Navier-Stokes Two-Phase Flow Solver, we have executed a scalability test on this 3D benchmark on a large number of processors, up to 1000. Table 3 and Figure 6 show the results for one iteration of the MO iteration. We observe a good scalability up to 200 processors, where we have the best trade-off between communication and computation time. Note that the graph of Figure 6 can be extrapolated. Indeed, our experience indicates, that

vu

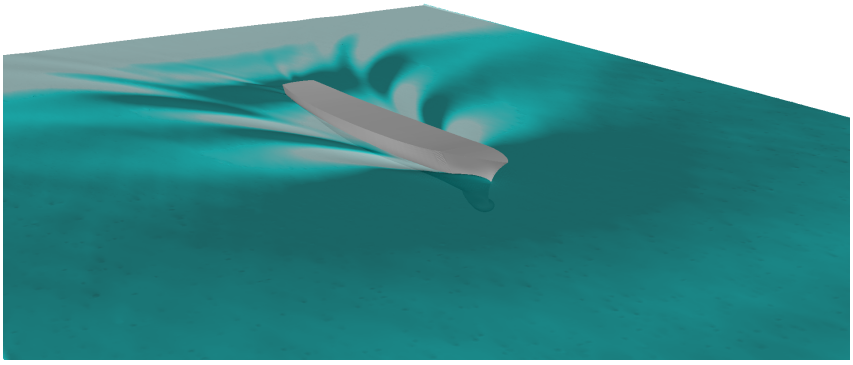

Figure 4: View of the Kelvin wake around steady DTMB 5512 at $Fr = 0.28$

the key parameter that limitates scalability is the number of nodes per processor. The number of elements by processors for the 1000 processors run is very few. Therefore, we expect that for a mesh with 5 times more nodes we could scale up to 1000 processors.

Processors Number	10	100	200	500	1000
Average Computation time per iteration (normalized by the 1000 processors values)	65.16	7.26	3.64	1.73	1
Acceleration	-	90	179	375	652
Elements Number per processors	1455390	145539	72769	29108	14554

Table 3: Parallel performance of two-phase flow solver on steady DTMB 5512 benchmark

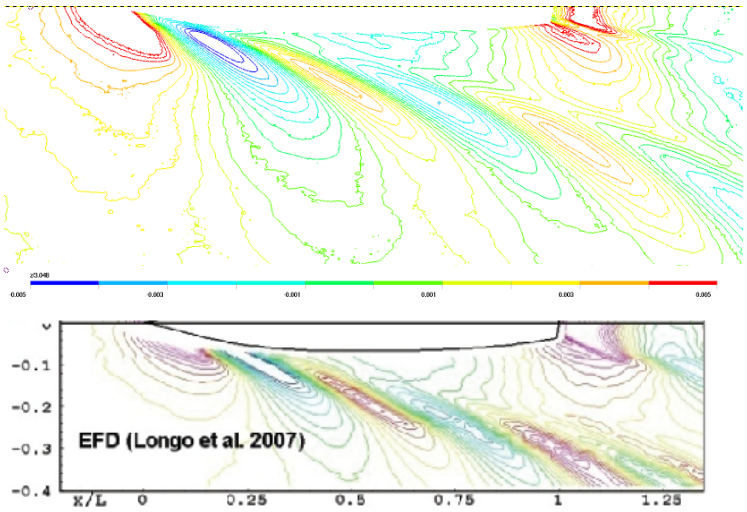

Figure 5: Wave elevation profile for DTMB5512 (Fr=0.28). Up: Level Set numerical results. Bottom: Experimental data [12] (contours from -5.10^{-3} to 5.10^{-3} with intervals of 5.10^{-4})

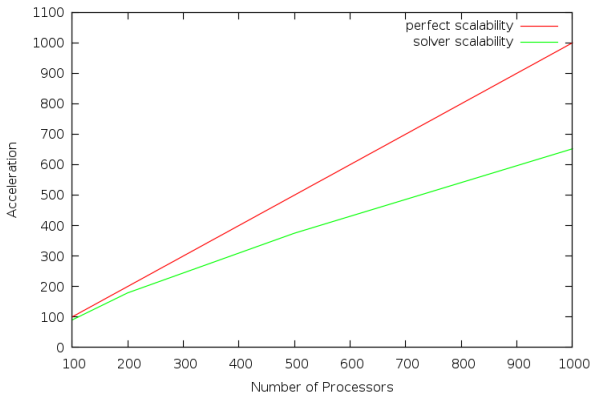

Figure 6: *Scalability test on steady DTMB 5512 benchmark*

8 CONCLUSIONS

In this paper, an extension of a massively parallel incompressible Navier-Stokes Eulerian solver [8] has been successfully applied to ship design process through a two-phase flow approach. The Finite Element discretization in space [6] along with the Level Set Method [24] to capture the interface motion allows the use of a flexible unstructured locally-refined mesh.

On the steady DTMB 5512 [9], the association of our two-phase solver with a Spalart-Allmaras turbulence modelling [27] and a Reichardt wall function approach was validated against experimental free surface elevation and forces. The combination of the LS method and the parallel fractional predictor-scheme proved high performance parallelization on this 3D benchmark.

Future work will include transient simulations, mesh adaptation and fluid-structure interactions.

REFERENCES

[1] Adalsteinsson, D., Sethian, J., A fast level set method for propagating interfaces. *J. Comput. Phys.* **118**, 269, 1995.

[2] Chorin, A., Numerical solution of the Navier-Stokes equations. *Math. Comput., 22:745762, 1968.32*

[3] Codina, R., A stabilized finite element method for generalized stationary incompressible flow . Computer Methods in Applied Mechanics and Engineering*, p-2681-2706, 2001.*

[4] J.H. Duncan. The breaking and non-breaking wave resistance of a two dimensional hydrofoil.J. Fluid Mech., 126:507-520, 1983.

[5] Golub, G. H. and Van Loan C. F. Matrix Computations. The Johns Hopkins University Press, 1996.

[6] Houzeaux, G., and Principe, J., A variational subgrid scale model for transient incompressible flows. Int. J. Comp. Fluid Dyn., 22(3):135-152, 2008.

[7] Houzeaux, G., Vazquez, M., Aubry, R., Cela, J. M., A massively parallel fractional step solver for incompressible flows. J. Comp. Phys., 228:6316-6332, 2009.

[8] Houzeaux, G., Aubry, R., Vazquez, M., Extension of fractional step techniques for incompressible ows: The preconditioned Orthomin(1) for the pressure Schur complement. J. Comp. Phys., 2009, submitted.

[9] Hydroscience and Engineering Ship Hydrodynamics Website. http://www.iihr.uiowa.edu/ shiphydro/efd.htm.

[10] The Italian Ship Model Basin. http://www.insean.it/

[11] Lesage, A-C, Alain, O., Dervieux, A., On level-set modelling of bi-fluid capillary flow. Int. J. Numer. Meth. Fluids*,* **53***, p 1297-1314, 2007.*

[12] Longo, J., Stern, F., Uncertainty Assessment for Towing Tank Tests With Example for Surface Combatant DTMB Model 5415, J. Ship Research Vol. 49, p 55-68, 2005.

[13] Lohner R, Yang C, Onate E. On the simulation of flows with violent free-surface motion. Computer Methods in Applied Mechanics and Engineering, 2006; 195:55975620.

[14] Olsson, E. and Kreiss, G., A conservative level set method for two phase flow. J. Comput. Phys. 210, p. 225-246, 2005.

[15] Osher, S. and Fedkiw, R., Level Set Methods and Dynamic Implicit Surfaces. Springer, 2002.

[16] Peng, D., Merrinam, B., Osher, S., Zhao, H-K., Kang, M., A PDE-based fast local level set method. J. Comput. Phys. **155***, 410, 1999.*

[17] Perot, J., B., An analysis of the fractional step method. J. Comput. Phys., 108:5158, 1993.

[18] Pilliod, Jr J.E., Puckett, E.G., Second-order accurate volume-of-fluid algorithms for tracking material interfaces. J. Comput. Phys., **199***(2), p. 465-502, 2004.*

[19] Quarteroni, A.,Saleri, F., and Veneziani, A., Factorization methods for the numerical approximation of navierstokes equations. Comp. Meth. Appl. Mech. Eng., 188:505526, 2000.

[20] Hinze J.O, "Turbulence" (2nd edn). McGraw-Hill: New York, 1975.

[21] Sethian, J.A, Level Set Methods and Fast Marching Methods. *Cambridge University Press, 1999.*

[22] Smolianski, A., Finite-Element/Level-Set/Operator-Splitting (FELSOS) Approach for Computing Two-Fluid Unsteady Flows with Free Moving Interfaces. Int. J. Numer. Meth. Fluids 48, p. 231-269, 2005.

[23] Sussman, M. and Pucket, E.G., A coupled level set and volume-of-fluid method for computing growth and collapse of vapor bubbles. J. Comput. Phys. 162, p. 301-337, 2000.

[24] M. Sussman, P. Smereka and S. Osher, "A level set approach for computing solutions to incompressible two-phase flow". J. Comput. Phys., Vol. 114, pp. 146-159, 1994.

[25] Sussman, M., Fatemi, E., Smereka, P. and Osher, S., An improved level set method for incompressible two-phase flows. Computers and Fluids, **27***, p. 663-680, 1998.*

[26] Sussman, M., Smereka, P. and Osher, S.,An level set approach for computing solutions to incompressible two-phase flow. J. Comput. Phys., **114***, p. 146-159, 1994.*

[27] P.R. Spalart and S.R. Allmaras, "A one-equation turbulence model for aerodynamic flows.," AIAA Paper, vol 93-2906, 1993.

[28] Temam, R., Sur lapproximation de la solution des équations de NavierStokes par la méthode des pas fractionaires (I). Arch. Rat. Mech. Anal., 32:135153,1969.

[29] Tezduyar TE, Behr M, Liou J. A new strategy for finite element computations involving moving boundaries and interfacesthe deforming-spatial-domain/spacetime procedure: I. The concept and the preliminary numerical tests. Computer Methods in Applied Mechanics and Engineering 1992; 94:339351.

[30] Tezduyar TE, Behr M, Mittal S, Liou J. A new strategy for finite element computations involving moving boundaries and interfacesthe deforming-spatial-domain/spacetime procedure: II. Computation of free-surface flows, two-liquid flows, and flows with drifting cylinders. Computer Methods in Applied Mechanics and Engineering 1992; 94:353371.

[31] Van Kan, J., A second-order accurate pressure correction scheme for viscous incompressible flow. SIAM J. Sci. Stat. Comput. Arch., 7:870891, 1986.