The Nepali Math. Sc. Report
Vol. 20 No. 1&2, 2002

Modeling Incompressible Navier–Stokes Flows by Least Squares Approximation

SUDARSHAN TIWARI* AND SANDRO MANSERVISI**

Abstract: In this paper the least squares particle method (LSQ) is used to model incompressible flows. We present a constrained least squares approximation for the reconstruction of the flow and show reliable and accurate simulations of the viscous Navier-Stokes equations. The corresponding incompressible limit can be implemented and its accuracy tested against numerical and analytical solutions. The incompressible Poiseuille and viscous multi-vortex flows are studied and compared with analytical solutions. Furthermore results for cavity flows at different Reynolds numbers are presented and discussed.

Keywords: Navier-Stokes equations, Particle method, Least squares approximation

AMS subject classification: 76D05, 76M28

1. Introduction

The study of incompressible fluid flows is one of the main field of computational fluid dynamics. This subject becomes even more appealing when one can see the incompressible flow as limit of the compressible, viscous Navier-Stokes equations. The aim of this paper is to show how the compressible, viscous Navier-

---

* Supported by DFG–Priority Research Programm : Analyssis and Numerics for Conservation Laws. tiwari@itwm.fhg. de. Fraunhofer Institut für Techno–und Wirtschaftmathematik, Gottlieb–Daimler–Strasse, Gebäude 49, 67663 Kaiserslautern, Germany.

** Suppoted by the European community under grant XCT–97–0117. Department of Mathematics, University of Kaiserslautern, 67663 Kaiserslautern, Germany. Actual address Nuuclear Engineering Deapartment, University of Bologna, Via dei Colli 16, 40136 Bologna, Italy sandro.manservisi@mail.ing.unibo.it

Stokes equations can be simulated by using the least squares method in the framework of particle method and the incompressible flow can be reproduced as limit of small Mach numbers.

The particle method has certain advantages over other methods. This method is a meshfree and a fully Lagrangian method and numerical solutions are computed over a moving grid defined by particles treating the geometry and the corresponding moving boundaries in a natural way. However the classical particle method, smoothed particle hydrodynamics (SPH) [11], is based on an integral interpolant with sufficiently smooth symmetric kernel function and has two fundamental problems: the approximation of derivatives of order larger than one and the implementation of the boundary conditions.

In the classical particle approach the approximation of derivatives near the boundary is not accurate. Alternatively, the approximation of derivatives in a grid free structure can be obtained by moving least squares methods [1, 3, 10]. In [10] it is shown that the moving least squares method gives a good approximation of the function and derivative near the boundary. Both of the approaches are similar to the finite difference discretization but show well known problems of instability and artificial viscosity should be introduced in order to stabilize the scheme. In [11] viscosity is introduced in the momentum and energy equations and in [10] an artificial viscous term is proposed for all the equations of the system. Both approaches do not give good approximations of the second order spatial derivative and therefore, they cannot compute efficiently the Navier-Stokes equations. In this paper, we approximate the first and second order derivatives by a constrained weighted least squares method and the natural Navier-Stokes viscous term is used. In this approach the solutions of the compressible Euler system can be obtained from the Navier-Stokes equations by letting the viscosity and heat conductivity tend to zero. In [15] the scheme for the $1D$ case is shown to be stable and numerical solutions converge to the Euler solutions when the number of particles tend to infinity and the viscosity and heat conductivity tend to zero.

The particle method should reproduce the results obtained by other well known methods with comparable accuracy. Many treatments of the boundary conditions are "ad hoc" implementations and cannot be reproduced easily. In this paper we propose to reconstruct the field over a fix grid and impose the boundary conditions over such a field. The fix grid can be used over the entire domain or only over part of the domain containing the boundary. We show that our method is consistent and it is a solid starting point for a moving particle method. In this paper we are discussing only the method over domains with fix boundaries leaving to further works the discussion of the rules necessary for consistent moving or adaptive particle grids. For further applications of the particle hydrodynamics to moving boundary problems we refer to [10].

The particle scheme is used to find the velocity and pressure fields and a new constrained least squares method is used to reconstruct the function and the derivatives. The use of the constrained least squares approximation allows us to compute with accuracy the second order space derivatives and the solution of the viscous Navier-Stokes equations. Since most of the numerical and analytical works are in the field of incompressible flows over bounded domains we apply this new method to incompressible flows. We solve basically the compressible Navier-Stokes equations in the low Mach number limit.

The paper is organized as follows. In §2 we introduce the compressible model and the incompressible limit. In §3 the constrained least squares method is described and the discrete set of equations are written. In §4 we present some numerical tests.

2. Model

Let Ω be an open bounded domain in $\mathbb{R}^s$ ($s = 1, 2, 3$) with boundary Γ. Let $\hat{\rho}, \hat{v}$ and $\hat{p}$ be the density, velocity and pressure fields representing the state variables. The compressible Navier-Stokes system in the Lagarangian form can be written as [6]

(2.1) $$\frac{D\hat{\rho}}{Dt} = -\hat{\rho}\nabla . \hat{v}$$

(2.2) $$\hat{\rho}\frac{D\hat{v}}{Dt} = -\nabla\hat{p} + \mu\vec{\nabla} . \tilde{\sigma}(\hat{v}),$$

where μ is the dynamic viscosity. By D/Dt we denote the Lagrangian derivative and by $\tilde{\sigma}$ the stress tensor $\tilde{\sigma}_{ij} = \frac{\partial\hat{v}_i}{\partial x_j} + \frac{\partial\hat{v}_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\ \nabla . \hat{v}$. The system (2.1–2.2) is closed by the state equation $\hat{p} = \hat{p}(\hat{\rho})$. In this paper we assume a simple linear state law $\hat{p} = A\hat{\rho} + B$ with $A = c^2$ and B constant where c is the characteristic sound speed. For an ideal compressible fluid a power law $\hat{p} = A\hat{\rho}^\gamma + B$ can also be used when A/γ is set to be equal to c^2. The Lagrangian form of the equations in (2.1–2.2) can be easily implemented over fix or free boundary domains. However, before solving problems in complex or moving geometries we would like to show that the method is accurate and it can be used to simulate incompressible flows.

The Navier-Stokes system for incompressible fluid flow is a different set of equations which can be written as

(2.3) $$\nabla . \hat{v} = 0$$

(2.4) $$\hat{\rho}\frac{D\hat{v}}{Dt} = -\nabla\hat{p} + \mu\vec{\nabla} . \tilde{\sigma}.$$

Under appropriate conditions over some data one expects that the system (2.1–2.2) converges to (2.3–2.4). We denote the nature of (2.1) is deeply different from (2.3) and we should expect the limit to be singular.

Numerical simulations of compressible flows at low Mach numbers is not easy due to the multiple time and length scales involved in the computation. In order to rewrite the system (2.1–2.2) in a more suitable form we use an asymptotic expansion which also gives a good insight into the solution behaviour of the compressible equations in the limit of vanishing Mach number. To cover small scale flow as well as the long-wave phenomena a single time scale is performed. Similar results can be found in literature in many forms and flavors. For details one can consult for example [8, 9, 13, 7].

Let ρ_{ref} , p_{ref} , V_{ref} be the reference state. We consider the state variables (ρ, $\vec{v}$, p) in the non-dimensional form, namely $\rho = \hat{\rho} / \rho_{ref}$, $\vec{v} = \hat{v} / V_{ref}$ and $p = \hat{p} / p_{ref}$. By taking a typical speed V_0 of the incompressible flow and the length scale L of the flow we set the time scale $\theta = L / V_{ref}$. We note that V_{ref} should be in some way related, but not necessarily equal, to V_0 or c. This leads to write the non-dimensional Navier-Stokes system, for the state variables ($\rho, p, \vec{v}$), as

$$\frac{D\rho}{Dt} = -\rho \nabla . \vec{v} \tag{2.5}$$

$$\rho \frac{D\vec{v}}{Dt} = -\frac{p_{ref}}{\rho_{ref} V_{ref}^2} \nabla p + \frac{\mu}{L V_{ref}\ \rho_{ref}} \nabla . \tilde{\sigma}. \tag{2.6}$$

Since we use the linear state law, we write $\hat{p} = c^2 (\hat{\rho} - \rho_{ref}) + V_0^2 \rho_{ref}$. In the limit of $\hat{\rho} \to \rho_{ref}$ we have $\hat{p} = V_0^2\ \rho_{ref}$, which is normally used as reference pressure in the incompressible flow. Now by solving (2.5–2.6) through an explicit method we should take into account the fact that we have quasi-incompressible flows and not fully divergent free flow. Let $\hat{\rho}_{hm}$ be the approximate average value for $\hat{\rho}$ then we can define $\Delta\hat{\rho}_h = \hat{\rho}_{hm} - \rho_{ref}$. In this numerical limit the reference pressure is defined by $\rho_{ref} = c^2 \Delta\rho_h + V_0^2\ \rho_{ref}$ which agrees with the incompressible reference pressure when $\Delta\hat{\rho}_h$ tends to zero. From the definition of p_{ref} we have $p_{ref} / \rho_{ref} = c^2 \Delta\hat{\rho}_h + V_0^2$. The state equation in the non-dimensional variable becomes $p = ((\rho - 1) + M_a^2) / (\Delta\rho_h + M_a^2)$, with the Mach number defined by $M_a^2 = V_0 / c^2$. The Mach number as a global parameter characterizing the non-dimensional limit is defined with respect to V_{ref} by $M = V_{ref} / (dp/d\rho)^{1/2} = \sqrt{\Delta\rho_h + M_a^2}$, which is equal to M_a in the limit of $\Delta\rho_h$ tending to zero. We note that $M \to 0$ implies both $M_a \to 0$ and $\Delta\rho_h \to 0$. If we set $V_{ref}^2 = p_{ref} / \rho_{ref}$ the system (2.5–2.6) becomes

(2.7) $$\frac{D\rho}{Dt} = -\rho\,\nabla.\vec{\upsilon}$$

(2.8) $$\rho\frac{D\vec{\upsilon}}{Dt} = -\nabla p + \frac{\mu}{LV_{ref}\,\rho_{ref}}\nabla\,.\,\widetilde{\sigma}$$

If M_a is small then V_{ref} is equal to $c\sqrt{\Delta\rho_h}$ and the time is scaled with a characteristic time for sound wave propagation. We have $M = \sqrt{\Delta\rho_h}$ and the non-dimensional velocity cannot be of order 0(1) in the limit of vanishing M_a.

If $\Delta\rho_h$ is small then V_{ref} is equal to V_0 and $M = M_a$. The time is scaled with a characteristic time for the incompressible flow propagation and the pressure term is singular (proportional to $1/M^2$). If one expands the variable p as $p = p_0 + M_a^2 p_2$ (see for examples [8,9,13,7] and references therein) there is no longer one single pressure term to influence the leading order velocity in the limit of low Mach number but a clean separation of different physical effects associated with these pressure terms. The leading term p_0 tends in the limit to be spatially homogeneous and acts as a thermodynamics variable satisfying the state equations. The second order term p_2 represents a balance between the inertial and viscous force and also guarantees the free divergence motion. In the vanishing limit this term should be decoupled completely from the total pressure and therefore from the state equation.

In spite of the fact that $M_a \to 0$ and $\Delta\rho_h \to 0$ are both singular limits the limit $M \to 0$ is nice if the ratio $\Delta\rho_h/M^2$ is approximately constant. The limit in $\Delta\rho_h$ is the most difficult to be imposed and in general a projection over a free divergence velocity field should be used [2, 12]. In this paper we use an explicit method to solve the Navier-Stockes system and therefore an asymptotic form of the (2.3–2.4) and the pressure state equation is appropriate. In (2.8) we take the limit $\Delta\rho_h \to 0$ and in the state equation the limit $M_a \to 0$. The system (2.3–2.4) and the pressure state equation become

(2.9) $$\frac{D\rho}{Dt} = -\rho\,\nabla.\vec{\upsilon}$$

(2.10) $$\frac{D\vec{\upsilon}}{Dt} = -\nabla p + \frac{1}{\mathrm{Re}}\nabla\,.\,\widetilde{\sigma}.$$

(2.11) $$p = \frac{(\rho-1)}{\delta}$$

where $\mathrm{Re} = L\rho_{ref}V_0/\mu$ is the Reynolds number and δ a positive small real number. The (2.10) is clearly the limit equation for M tending to zero but the (2.11) should be understood in the limit of small δ. We note that the (2.9–2.10) has the same form of the compressible system in (2.1–2.2) and therefore suitable for a Lagrangian particle method.

We would like to remark that (2.11) is not anymore a state equation but an equation for the incompressible pressure which is completely decoupled from the compressible state equation. The constant δ determines the pressure in agreement with the variable ρ which is not anymore representing the real density but a sort of error in the divergence field.

3. LSQ–particle Discretization

3.1. Least Squares Approximation of the derivatives

The least squares method has been used to approximate the first order space derivatives and solve the compressible Euler equations in fix and moving geometrics (see for example [4, 5]). The aim of this paper is to approximate the full Navier - Stokes equations and therefore both first and second order space derivatives. The main advantage of the least squares methods is that it is very general and can be applied to very irregular moving geometries. The idea is to substitute the smoothing functions used to interpolate particle solutions with a very general least squares interpolant which can cope with a large class of mesh configurations.

Let $f(t,\vec{x})$ be a scalar function and $f_i(t)$ its values at $\vec{x}_i$ for $i = 1, 2, \ldots, N$ and time t. Consider the problem to approximate the function and the spatial derivatives of the function $f(t,\vec{x})$ at $\vec{x}$ in terms of the values of a set of neighboring points. In order to limit the number of points we associate a weight function $w = w(\vec{x}_i - \vec{x}\,; h))$ with small compact support, where h determines the size of the support. In the classical smoothed particle hydrodynamics method, h is known as smoothing length. The weight function can be quite arbitrary but in our computations, we consider a Gaussian weight function in the following form

$$w = w(\vec{x}_i - \vec{x}\,; h)) = \begin{cases} \exp\left(-\alpha \frac{\|\vec{x}_i - \vec{x}\|^2}{h^2}\right), & \text{if } \frac{\|\vec{x}_i - \vec{x}\|^2}{h} \le 1 \\ 0, & \text{else,} \end{cases}$$

with α a positive constant. The smoothing length defines a set of neighboring particles around $\vec{x}$. Let $P(\vec{x}) = \{\vec{x}_i : i = 1, 2, \ldots, n\}$ be the set of n neighboring points of $\vec{x}$. The distribution of neighboring points needs not to be uniform and it can be quite arbitrary. For consistency reasons some obvious restrictions are required, namely for example the particles should not be on the same line.

We approximate the function $f(t,\vec{x})$ by $f_h\ (t,\vec{x})$ as $f_h\ (t,\vec{x}) = \sum_{i=1}^{N} f_i(t)\ \phi_h\ (\vec{x}_i, x)$, where the shape function $\phi_h(\vec{x}_i, \vec{x})$ is computed at each point $\vec{x}$ by the least squares method over its own compact support. It is important to stress that this expression is consistent only if the function ϕ_h is 1 at $\vec{x}_i$, namely $\phi_h(\vec{x}_i,\vec{x}_j)=\delta_{ij}$ for all $i, j = 1, 2, \ldots, N$.

The approximation of the first and second order derivatives can be computed directly from $f_h(t,\vec{x})$ or directly by using the least squares method. The first method is known in literature as moving least squares method [3, 10]. Usually the function $f_h(t,\vec{x})$ and its derivatives $f_{kh}(t,\vec{x})$ are not smooth enough to be differentiable and therefore the second order derivatives cannot properly computed.

In this paper we approximate the derivatives $\partial f(t,\vec{x})/\partial x_k$ by $f_{kh}(t,\vec{x}) = \sum_{i=1}^{N} f_i(t)\ \eta_{kh}\ (\vec{x}_i,\vec{x})$ for k = 1,2,3, where $\eta_{kh}(\vec{x}_i,\vec{x})$ is directly computed by the least squares interpolation. In a similar manner we define the approximation for the second order derivatives $\partial^2 f(t,\vec{x})\,/\,\partial x_l\,\partial x_k$ by $f_{klh}(t,\vec{x}) = \sum_{i=1}^{N} f_i(t)\psi_{kh}\ (\vec{x}_i,\vec{x})$ for k = 1, 2, 3. The determination of the functions $f_h(t,\vec{x})$, $f_{kh}(t,\vec{x})$, and $f_{klh}(t,\vec{x})$ $(=f_{lkh}\ (t,\vec{x}))$ for k, l = 1, 2, 3 can be computed easily and accurately by using the Taylor series expansion and the least squares approximation. We write a Taylor's expansion around the point $\vec{x}$ with unknown coefficients and then compute these coefficients by minimizing a weighted error over the neighboring points. The optimization is constrained to satisfy $\phi_h(\vec{x}_1,\vec{x}_1)=1$ where $\vec{x}_1$ is the closest point, namely the approximation must interpolate the closest point.

In order to approximate the function and its derivatives at $\vec{x}$ by using a quadratic approximation through the n neighboring points sorted with respect to its distance from $\vec{x}$ we let

$$f(t,\vec{x}_i) = f_h(t,\vec{x}) + \sum_{k=1}^{3} f_{kh}(t,\vec{x})\,(x_{ki}-x_k)$$
$$+\frac{1}{2}\sum_{k,l=1}^{3} f_{lkh}(t,\vec{x})\,(x_{ki}-x_k)\,(x_{li}-x_k)+e_i,$$

where e_i is the error in the Taylor's expansion at the point $\vec{x}_i$. The unknowns f_h, f_{kh} and f_{klh} for k, l = 1,2,3 are computed by minimizing the error e_i for $i = 2, 3, \ldots, n$ and setting the constraint $e_1 = 0$. Our method to solve this constrained least squares problem is straightforward. By subtracting the first equation with $e_1 = 0$ to all the other equations the system can be written as $\vec{e} = M\vec{a} - \vec{b}$, where

$$M=\begin{pmatrix} \Delta x1_2 & \Delta x2_2 & \Delta x3_2 & \Delta x11_2 & \Delta x12_2 & \Delta x13_2 & \Delta x22_2 & \Delta x23_2 & \Delta x33_2 \\ \Delta x1_3 & \Delta x2_3 & \Delta x3_3 & \Delta x11_3 & \Delta x12_3 & \Delta x13_3 & \Delta x22_3 & \Delta x23_3 & \Delta x33_3 \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ \Delta x1_n & \Delta x2_n & \Delta x3_n & \Delta x11_n & \Delta x12_n & \Delta x13_n & \Delta x22_n & \Delta x23_n & \Delta x33_n \end{pmatrix},$$

where $\vec{a} = [f_{1h}, f_{2h}, f_{3h}, f_{11h}, f_{12h}, f_{13h}, f_{22h}, f_{23h}, f_{33h}]^T$, $\vec{b} = [f_2 - f_1, (f_3 - f_1), \ldots, f_n - f_1]^T$, $\vec{e} = [e_2, e_3, \ldots, e_n]^T$. The symbol $\Delta x k_i$ denotes $x_{ki} - x_{k1}, \Delta x k l_i$ denotes $(x_{ki} - x_k)(x_{li} - x_l)$ and $\Delta x k k_i$ the quantity $(x_{ki} - x_k)(x_{ki} - x_l)/2$ for $k,\ l = 1,2,3$ and $i = 2,3, \ldots, n$.

For $n > 9$, this system is over-determined for the nine unknowns f_{kh} and f_{klh} for $k, l = 1,2,3$.

The unknowns $\vec{a}$ are obtained from a weighted least squares method by minimizing the quadratic form $J = \Sigma_{i=1}^{n} w_i e_i^2$. The above equations can be expressed in the form $J = (M\vec{a} - \vec{b})^T\ W(M\vec{a} - \vec{b})$ where $W = \delta_{ij} w_i$. The minimization of J formally yields $\vec{a} = (M^T W M)^{-1}(M^T W)\vec{b}$. Now from the equation for the closest point x_1 we can compute the value of $f_h(t, \vec{x})$ at $\vec{x}$ as

$$f_h(t,\vec{x}) = f(t,\vec{x}_1) - \sum_{k=1}^{3} f_{kh}(t,\vec{x})(x_{1l} - x_k) - \frac{1}{2}\sum_{k,l=1}^{3} f_{lkh}(t,\vec{x})(x_{k1} - x_k)(x_{l1} - x_k)$$

since f_{kh} and f_{klh} for $k, l = 1,2,3$ are now known..

The solution of the constrained least squares problem is straightforward and more sophisticated techniques can be used. For example minimization or singular decomposition techniques can be very helpful to determine efficiently the unknowns.

We note that if the approximation is computed at $\vec{x}_i$ we have $f_h(t, \vec{x}_i) = f_i(t)$ which implies $\phi(\vec{x}_i, \vec{x}_j) = \delta_{ij}$ for all $i, j = 1,2, \ldots, N$. Also we note that if the weight function is chosen in a suitable form then the constraint $\phi(\vec{x}_i, \vec{x}_j) = \delta_{ij}$ can be approximated very closely performing the unconstrained least squares minimization over all the n equations.

3.2. LSQ-particle discretization

The idea behind the least squares particle method is to approximate a space-time function by mans of an expansion which is represented by scaled and displaced approximate delta functions at the particle position.

Let $f(t, \vec{x})$ be a scalar function and $f_i(t)$ be the set of its values at the particle points $\vec{x}_i$ for $i = 1, 2, \ldots, N$ and time t. We approximate the function $f(t,\vec{x}) = \Pi f_h(t,\vec{x}) = \Sigma_{i=1}^{N} f_i(t)\, \phi(\vec{x}_i, \vec{x})$, its derivatives f_k and f_{kl} as $f_{kh}(t, \vec{x}; \vec{z}) = \Pi_k f_h(t,\vec{x}) = \Sigma_{i=1}^{N} f_i(t)\ \eta_k(\vec{x}_i, \vec{x})$ and $f_{klh}(t, \vec{x}; \vec{z}) = \Pi_{lk} f_h(t,\vec{x}) = \Sigma_{i=1}^{N} f_i(t)\, \psi_{kl}(\vec{x}_i, \vec{x})$ respectively. The functions ϕ, η_k and ψ_{kl} are computed at each point through the constrained least squares approximation described in the previous section by using the neighboring points over their compact support. The operators Π, Π_k and Π_{lk} for

$k, l = 1, 2, 3$ are well defined and give the values of the function and its derivatives as a linear combination of the neighboring points.

We can summarize some abstract properties for Π which allow us to write the discrete particle approximation. The operator Π satisfies the following properties:

1) the operator Π is linear and the approximation depends linearly from the particle point values ;
2) the approximation obtained by applying the least square method is consistent and the evaluation at the particle points gives the interpolating value. Therefore $\Pi f(t,\vec{x}_i) = f_i(t) = f(t,\vec{x})$ for all $i = 1, 2, \ldots, N$.
3) From the above formalism we have $\Pi f_k\ (t,\vec{x}) = \Pi_k f(t,\vec{x})$ and $\Pi f_{kl}(t,\vec{x}) = \Pi_{kl} f(t,\vec{x})$ for $k,l = 1, 2, 3$.

Consider the system in (2.9–2.11), by introducing the least square approximation and by using the properties in (2–3) we have

$$\frac{D\rho_i(t)}{Dt} = -\rho_i(t)\Pi\nabla \cdot \vec{v}(t,\vec{x}_i) \tag{3.12}$$

$$\rho_i(t)\frac{D\vec{v}_i(t)}{Dt} = -\Pi\nabla p(t,\vec{x}_i) + \frac{1}{\mathrm{Re}}\Pi\nabla \cdot \vec{\sigma}(\vec{v}(t,\vec{x}_i)) \tag{3.13}$$

$$p_i(t) = \frac{(\rho_i(t)-1)}{\delta} \tag{3.14}$$

at $\vec{x}_i$ for all $i = 1, 2, \ldots, N$. Since the partial derivatives on the *rhs* are approximated by the operators Π, Π_k, Π_{lk} for $k,l = 1, 2, 3$ the system of partial differential equations reduces to a time dependent system of ordinary differential equations.

In addition to the Navier-Stokes system the equations that determine the particle positions should be included as

$$\frac{d\vec{x}_i}{dt} = \beta\,\vec{v}_i, \text{ for } \quad i = 1, \ldots, N, \tag{3.15}$$

with $0 \le \beta \le 1$. For the case $\beta = 1$ each particle moves with its own velocity along the streamlines ; if $\beta < 1$ the particle moves with reduced velocity field and if $\beta = 0$ the motion is considered with respect to a fix particle grid. It is clear that if β is not equal to 1 the fields must be reconstructed over the points of interest. This approach allows a great variety of possibilities: the particles can be traced along their streamlines or traced at fix positions. For problems over fix domains the use of moving grids could be not effective since the grid may deform and the solution may loose accuracy. Furthermore the implementation of the boundary conditions which is easy in the Euler formulation cannot be done efficiently in the Lagrangian formulation. Our approach is to apply the boundary conditions always over a fix grid of particles. This grid must cover fix boundaries but can also be extended to cover all the domain that does not have free surfaces.

After the approximation of the spatial derivatives in (3.12–3.15) these equations reduce to a system of ordinary differential equations. This system can be solved by a simple integrations scheme. One can use the explicit Euler scheme, but it requires very small time step. The simple explicit forward Euler scheme is in some cases insufficient to give satisfactory results and, if possible, higher order methods should be used. Here a two Runge-Kutta time steps is proposed which is sufficient for many of the tests proposed in the next section [14].

Let $\mathbf{y}_i = [\vec{x}_i, \rho_i, \vec{v}_i]$ and

$$\mathbf{F}_i(t, \mathbf{y}, \nabla\mathbf{y}, \Delta\mathbf{y}) = \begin{bmatrix} \vec{v}_i \\ -\rho_i \Pi\nabla . \vec{v}(t, \vec{x}_i) \\ -\Pi\nabla . p(t, \vec{x}_i) + \frac{1}{\mathrm{Re}} \Pi\widetilde{\sigma}(\vec{v}(t, \vec{x}_i)) \end{bmatrix}$$

for $i = 1, 2, \ldots, N$. In agreement with the notation introduced above the discrete system can be rewritten in a compact form as

$$\frac{d\mathbf{y}_i}{dt} = \mathbf{F}_i(t, \mathbf{y}, \nabla\mathbf{y}, \Delta\mathbf{y}), \tag{3.16}$$

for $i = 1, 2, \ldots, N$, where $\mathbf{F}_i$ denote the discrete approximation of the right hand sides in (3.12-3.13) and (3.15).

4. Numerical Tests

In this section we present some numerical tests. We always consider discretizations over bounded domains with N particles at $\vec{x}_i$ for $i = 1, 2, \ldots, N$ and constant time step Δt.

In this section we denote by f_i^m the values $f(t, \vec{x}_i)$ with $t = m\Delta t$ for $m = 0, 1, \ldots$. The discrete form of the Navier-Stokes equations are the discrete version of the asymptotic equations in (3.12-3.15) for Runge-Kutta two time steps, namely we solve

$$\rho_i^{m+\frac{1}{2}} = \rho_i^m - \frac{\Delta t}{2}\, \rho_i^m\, \Pi\nabla . \vec{v}_i^m \tag{4.17}$$

$$\vec{v}_i^{m+\frac{1}{2}} = \vec{v}_i^m - \frac{\Delta t}{2\rho_i^m} \left(\Pi\nabla p_i^m + \frac{1}{\mathrm{Re}} \Pi\nabla . \widetilde{\sigma}(\vec{v}_i^m) \right) \tag{4.18}$$

$$p_i^{m+\frac{1}{2}} = \frac{(\rho_i^{m\frac{1}{2}} - 1)}{\delta} \tag{4.19}$$

(4.20) $$\bar{x}_i^{m+\frac{1}{2}} = \bar{x}_i^m + \frac{\Delta t}{2}\,\bar{v}_i^m$$

(4.21) $$\rho_i^{m+\frac{1}{2}} = \rho_i^m - \Delta t\ \rho_i^{m+\frac{1}{2}}\,\Pi\nabla\,.\,\bar{v}_i^{m+\frac{1}{2}}$$

(4.22) $$\bar{v}_i^{m+\frac{1}{2}} = \bar{v}_i^m - \frac{\Delta t}{\rho_i^{m+\frac{1}{2}}}\left(\Pi\nabla p_i^{m+\frac{1}{2}} + \frac{1}{\mathrm{Re}}\Pi\nabla\,.\,\tilde{\sigma}(\bar{v}_i^{m+\frac{1}{2}})\right)$$

(4.23) $$p_i^{m+1} = \frac{(\rho_i^{m+1} - 1)}{\delta}$$

(4.24) $$x_i^{m+1} = \bar{x}_i^m + \Delta t\bar{v}_i^{m+\frac{1}{2}}$$

for $i = 1, 2, \ldots, N$ and $m = 0, 1, 2, \ldots$, where the initial conditions are given by $(\rho_i^0, \bar{v}_i^0)$ for all $i = 1, 2, \ldots, N$ and δ small positive number.

Many strategies can be adopted. One possibility is to perform two time steps along streamlines and then interpolate the solution over a more regular grid of points. This is convenient in order to control the grid points and keep them regular. In this case the boundary conditions can be imposed simply fixing the velocity at the boundary. Another possibility is to move continuously the particles over the streamlines with its own or reduced velocity. The boundary conditions must be imposed through special boundary particles which are sitting over a boundary fix grid.

First we propose Poiseuille flow in order to test the constrained least squares method where we compute the viscous and body forces. In particular this flow tests the approximation of the second derivatives in space. However in these two tests the pressure does not play any role and the incompressibility constraint acts in a straightforward manner. Then we test the solution for moderate distribution of pressure against analytical solutions. Finally we test the driven cavity flow against the corresponding finite element approximation where the distribution of pressure is not trivial as in the previous cases.

4.1 Poiseuille Flow

The first test case is a stationary forced flow through a channel between two infinite parallel plates. The solution $\bar{v} = (u, v)$ of this simple flow can be written in series form as

(4.25) $$u(\bar{x}, t) = \frac{F}{2v}\,y(y - L) +$$

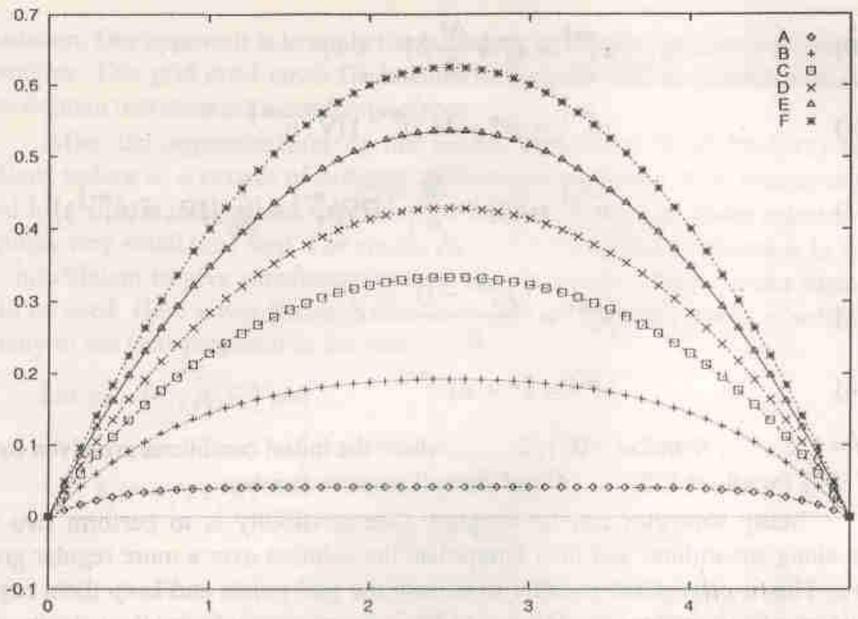


Figure 1: Test 1. Exact flow and solution for Poiseuille flow

$$+\frac{4FL^2}{\upsilon\pi^3}\sum_{n=0}^{\infty}\frac{1}{(2n+1)^3}\sin\left(\frac{\pi y}{L}(2n+1)\right)\exp\left(-\upsilon\frac{(2n+1)^2\pi^2}{L^2}t\right),$$

$$(\bar{x},t)=0,$$

where L is the width of the channel and F the force. We note that, we have to add the body force F in the momentum equation.

The test was performed with $L = 5$, $F = 1$ and $Re = 1$. Fig. 1 shows the exact and computed solution. We note that the solution obtained by the particle method approximates closely the flow. The solution has been computed both reconstructing the solution over a fix point grid at each time step or letting the particle flow along streamlines. In both cases the matching is excellent.

In Fig. 1 we plot the exact and particle solutions obtained by reconstructing the velocity field over the initial regular particle distribution at each time step. The solutions are plotted at $t = 0.041$ (A), $t = 0.201$ (B), $t = 0.401$ (C), $t = 0.601$ (D) $t = 1.001$ (E), and $t = \infty$ (F) respectively. The limit in δ does not present particular problems and the fact that the pressure is constant allows the viscous term to dominate the pressure term even for very small values of δ.

4.2. Flow in a Square

The Poiseuille flow is one-dimensional flow and do not produce variations in dynamics pressure. In those cases the incompressible limit defined by δ tending to zero cannot be tested in a proper way. In this subsection we present a test where the pressure is changed in agreement with a smooth quadratic distribution. We test the solution against the analytical solution for a flow driven by a given force in the square $(0,1) \times (0,1)$ with homogeneous Dirichlet boundary conditions. Let $\vec{v}_d = (u_d, v_d)$ be the desired velocity defined by

$$(4.26) \qquad u_d = \frac{d\phi(x,y)}{dy} \qquad v_d = -\frac{d\phi(x,y)}{dx}$$

where $\phi(x, y)$ is $\phi(x, y) = \Phi(x)\,\Phi(y)$ and $\Phi(z)$ is

$$\Phi(z) = (1-\cos(4\pi z))\,(1-z^2).$$

The pressure is given by $p_d = 10^5\ (x^2 - .25x)$. For given $(\vec{v}_d, p_d)$, the corresponding body force is

$$(4.27) \qquad \vec{F} = (\vec{v}_d\ .\ \vec{\nabla})\vec{v}_d + \frac{\nabla p_d}{\rho} - \frac{1}{\mathrm{Re}}\nabla\tilde{\sigma}(\vec{v}_0)$$

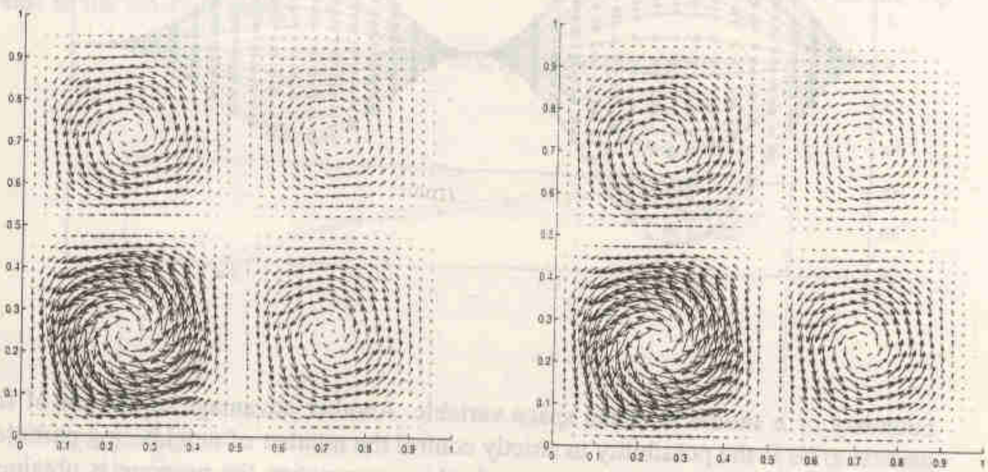
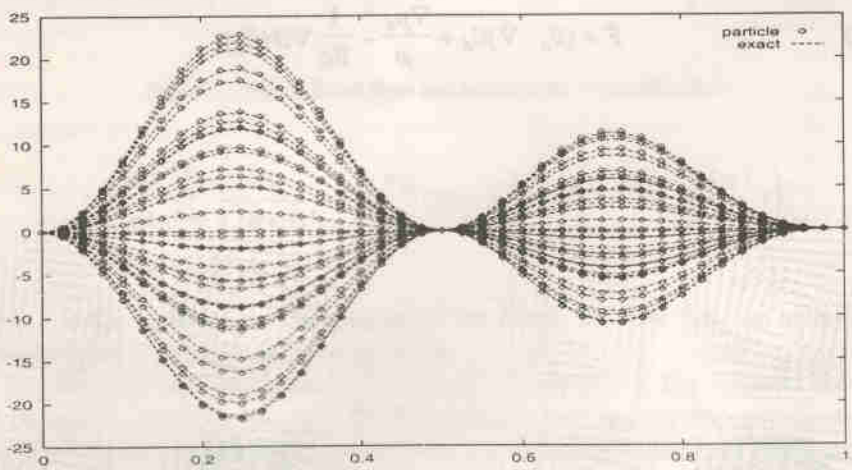


Figure 2: Test 2. Exact flow (right) and solution (left)

The flow consists of four vortices rotating in different parts of the domain. In order to obtain this solution we use a fix particle grid generated at the initial time. The boundary conditions are imposed simply by setting zero velocity at the boundary and reconstructing the fields at each time step. Since the flow is extremely complex the grid has been improved to 41×41 particles.

In Figs. 2–3 we have the solution obtained by the particle method against the exact expression. We can see that the solutions match perfectly: the u component is shown in Fig.3. In these figures all the 41 sections are plotted along the x-axis showing a high degree of symmetry and accuracy. The pressure, which is assumed to be parabolic is matched almost perfectly. Again the smoothing length h is 2.5 times the characteristics mesh length Δx. The ideal value for h over a regular grid should be evaluated on the basis of the number of neighboring particles necessary to compute the unknowns quantities. Usually $h = 1.5\ \Delta x$ is sufficient in the interior of the domain but not on the boundary, where the number of neighboring particles available is reduced. Therefore the values of h is dictated by the topology of the



boundary or h should be taken space variable. Another advantage of the use of fix particle grids is the possibility to strictly control the number of neighboring particles and therefore the approximation error. In this computation the pressure is obtained by $\delta = 10^9$ and density error less than 0.1%.

4.3. Driven cavity flow

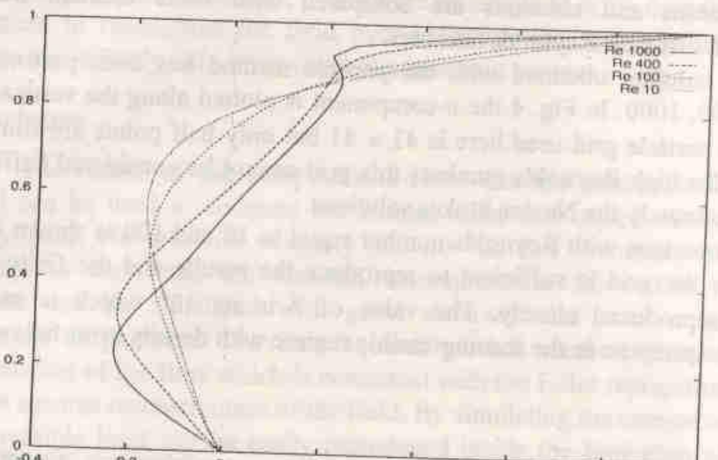


Figure 4: Test 3. U-component along the y-axis at $x = 0.5$.

The flow in a cavity driven by the velocity on the top has become a popular example for testing and comparing numerical methods. The velocity $\bar{u}$ on the top side of the cavity is chosen as

$$v(x,1) = 0 \qquad u(x,1) = 16x^2(1-x)^2,$$

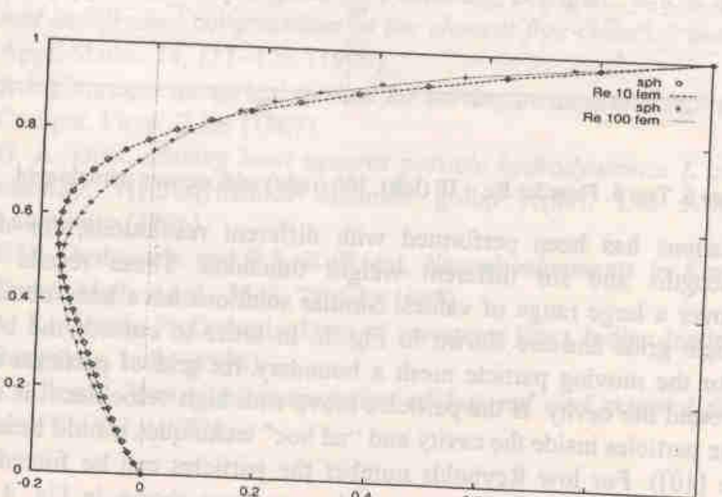


Figure 5: Test 3. U-component along the y-axis at $x = 0.5$.

with homogeneous Dirichlet boundary conditions over the rest of the boundary. Computations have been performed with the iterative method described in the previous sections and solutions are compared with finite element numerical solutions with different Reynolds numbers.

The solution obtained with the particle method has been performed for Re 10,100,400, 1000. In Fig. 4 the u-component is plotted along the vertical line at $x = 0.5$. The particle grid used here is 41×41 but only half points are shown. We remark that for high Reynolds numbers this grid cannot be considered sufficient to reproduce accurately the Navier-Stokes solutions.

Comparison with Reynolds number equal to 10 and 100 is shown in Fig.5. In this range the grid is sufficient to reproduce the results and the finite element solution is reproduced closely. The value of δ is set 10^4 which is enough to reproduce the pressure in the incompressible regime with density error below 0.1%.

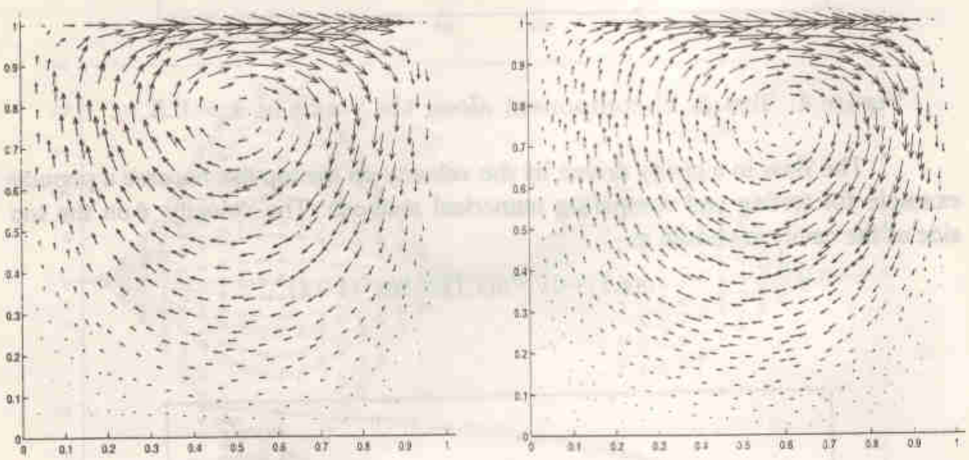


Figure 6. Test 3. Flow for Re = 10 (left), 100 (right) with moving particle grid.

The computations has been performed with different resolutions, for different smoothing lengths and for different weight functions. These results can be reproduced over a large range of values. Similar solutions have been obtained for moving particle grids and are shown in Fig. 6. In order to enforce the boundary conditions for the moving particle mesh a boundary fix grid of particles must be generated around the cavity. If the particles move with high velocities it is not easy to contain the particles inside the cavity and "ad hoc" techniques should be used (see for example [10]). For low Reynolds number the particles can be forced to stay inside the domain and the results are very close to those shown in Fig. 4 and the velocity profiles in Fig. 5 are matched by the solutions shown in Fig. 6. However for higher Reynolds a robust "ad hod" strategy is necessary to keep the particles inside the boundary. For these reasons, if flows are investigated in fix domains, it is always convenient to reconstruct the field over a "controlled" particle grid and let the particles move freely only if moving boundaries are present.

5. Conclusion

The results of the computations show that the constrained least squares method can be used to compute the second order derivatives and reconstruct the velocity field. No artificial viscosity and no "ad hoc" boundary conditions are necessary to reproduce the viscous, incompressible flow. The Navier-Stokes equations can be simulated with good accuracy and over a quite arbitrary distribution of particles. The least squares particle method gives a Lagrangian representation of the flow which is consistent with the Euler representation given by the least squares reconstruction of the field. By simulating the compressible flow the incompressible limit can be easily reproduced inside the limitation of the explicit scheme. Within the proposed formulation this limitation can only be improved by the development of an implicit or projection scheme. An improvement in this direction can be reached by enforcing the incompressibility constraint directly into the constrained least square approximation of the field. Also these scheme can be easily extended to problems with non-isothermal flow, complex geoemtries and moving boundaries.

REFERENCES

[1] T. Belytschko, Y. Krongauz, M. Flemming, D.Organ, W.K.S. Liu, *Smoothing and accelerated computations in the element free Galerkin method*; J. Comp. Appl. Maths. 74, 111–126, (1996).

[2] A.J. Chorin, *A numerical method for solving incompressible flow problems*; J. Comput. Phys., 2,12, (1967).

[3] G. A. Dilts, *Moving least squares particle hydrodynamics I, consistency and stability*; Hydrodynamics methods group report, Los Alamos National Laboratory (1996)

[4] S.M. Deshpande and P.S. Kulkarni, *New developments in Kinetics schemes*; Comp. Math. Appl., 35, 1 75–93, (1998)

[5] A. K. Ghosh, Ph.D thesis; Dept of aerospace Eng., Indian Institute of Science Bangalore, India (1996).

[6] C. Hirsch, *Numerical computation of Internal and external flows*; Springer eds., Vol 1,2, (1992).

[7] R. Klein, *Semi-implicit extension of a Godunov-type scheme based on low Mach number asymptotic* I: one dimensional flow; J. Comp. Phy., 121, 213–237 (1995).

[8] S. Klainerman and A. Majda, *Compressible and Incompressible fluids*; Comm. On Pure and Appl. Math., 35, 629–651 (1982).

[9] S. Klainerman and A. Majda, *Singular limit of quasi-linear hyperbolic systems with large parameters and the incompressible limit of incompressible fluids*; Comm. On Pure and Appl. To appear.

[10] J. Kuhnert, *General smoothed particle hydrodynamics*; Ph.D. thesis, Kaiserslautern University, Germany (1999)

[11] J.J. Monaghan and R.A. Gingold, *Shock Simulation by particle method SPH*; J. Comp. Phys., 52. 374–389. (1983)

[12] T. Schneider, N.Botta, K.J. Geratz and R.Klein, *Extension of finite volume compressible flow solvers to multi-dimensional, variable density Zero Mach number flows*; J. Comp. Phys., 155, 248–286 (1999).

[13] S. Schochet, *Asymptotics for symmetric hyperbolic systems with large parameters*; J. Diff. Eq., 75, 1–27, (1998).

[14] J. Stoer and R. Bulirsch, *Introduction to numerical analysis*; Spring-Verlag, New York (1980).

[15] S.Tiwari, *A least squares SPH approach for compressible flows*, International series of numerical mathematics Vol. 141, ed. By H. Freistuchler, G. Warnecke, Birkhaueser, 2000.

SUDARSHAN TIWARI
Franuhofer Institute für Techno–und Wirtschaftmathmematik,
Gottlieb–Daimier–Strasse,
Gebäude Gebaude 49, 67663
Kaiserslautern,
Germany

SANDRO MANSERVISI
Department of Mathematics,
University of Kaiserslautern, 67663,
Germany