Accepted Manuscript

High-Order Eulerian Incompressible Smoothed Particle Hydrodynamics with Transition to Lagrangian Free-Surface Motion

S.J. Lind, P.K. Stansby

PII: S0021-9991(16)30404-1
DOI: http://dx.doi.org/10.1016/j.jcp.2016.08.047
Reference: YJCPH 6803

To appear in: Journal of Computational Physics

Received date: 18 August 2015
Revised date: 24 August 2016
Accepted date: 29 August 2016

Please cite this article in press as: S.J. Lind, P.K. Stansby, High-Order Eulerian Incompressible Smoothed Particle Hydrodynamics with Transition to Lagrangian Free-Surface Motion, J. Comput. Phys. (2016), http://dx.doi.org/10.1016/j.jcp.2016.08.047

This is a PDF file of an unedited manuscript that has been accepted for publication. As a service to our customers we are providing this early version of the manuscript. The manuscript will undergo copyediting, typesetting, and review of the resulting proof before it is published in its final form. Please note that during the production process errors may be discovered which could affect the content, and all legal disclaimers that apply to the journal pertain.
High-Order Eulerian Incompressible Smoothed Particle Hydrodynamics with Transition to Lagrangian Free-Surface Motion

S. J. Lind and P. K. Stansby

School of Mechanical, Aerospace and Civil Engineering, The University of Manchester, Manchester, UK, M13 9PL

Abstract

The incompressible Smoothed Particle Hydrodynamics (ISPH) method is derived in Eulerian form with high-order smoothing kernels to provide increased accuracy for a range of steady and transient internal flows. Periodic transient flows, in particular, demonstrate high-order convergence and accuracies approaching, for example, spectral mesh-based methods. The improved accuracies are achieved through new high-order Gaussian kernels applied over regular particle distributions with time stepping formally up to 2nd order for transient flows. The approach can be easily extended to model free surface flows by merging from Eulerian to Lagrangian regions in an Arbitrary-Lagrangian-Eulerian (ALE) fashion, and a demonstration with periodic wave propagation is presented. In the long term, it is envisaged that the method will greatly increase the accuracy and efficiency of SPH methods, while retaining the flexibility of SPH in modelling free surface and multiphase flows.

Keywords: High-order SPH, incompressible SPH, Eulerian SPH, ALE-ISPH

1. Introduction

The numerical method Smoothed Particle Hydrodynamics (SPH) originated in astrophysics [42] and has since been the subject of considerable research activity largely because of its potential for simulating highly transient free-surface flows. It is traditionally a Lagrangian particle method, and the benefits of a fully Lagrangian approach for fluid flows have been so attractive that consideration of an Eulerian SPH (ESPH) counterpart has gone largely unaddressed. There is only one study involving ESPH known to the authors [48], where fixed particles (with standard 2nd order kernels) are used in an immersed boundary formulation for SPH. This paper presents a more comprehensive investigation into Eulerian SPH (ESPH) for the incompressible flow of a Newtonian fluid, with focus on the amenability of the method to high-order spatial solutions and the possibilities for coupled free-surface flow modelling. ESPH remains meshless in the sense that there are no strict requirements on node (particle) connectivity or ordering. Therefore, the method remains closely related to other meshless methods (e.g. Partition of Unity [43], Moving Least Squares [35], Element-Free Galerkin methods [7]), with the essential differences appearing through the choice of kernel or shape function, and the method of solution (collocation or Galerkin-type) [6]. Specifically, Eulerian SPH and Lagrangian SPH solve for a strong solution of the governing equations using collocation through particle positions. In contrast, Element-Free Galerkin (EFG) methods [7] and similar methods (e.g. meshless Petrov-Galerkin [2]) solve the associated weak form. Quadrature for the weak form may be done over background integration cells [21] or nodal integration [5] (like SPH), and enforcement of essential boundary conditions may be achieved through the use of Lagrange multipliers in the weak formulation (e.g. [34]), or through a natural coupling with finite elements [33]. When solving the linear system associated with Poisson's equation, a common feature of incompressible solvers, the test functions in a Galerkin approach yield a symmetric system amenable to efficient methods of solution, something not generally applicable in (incompressible) SPH methods. To date, EFG methods have been applied in a number of areas, notably fracture mechanics and crack propagation (e.g. [9, 8]), where continuity and differentiability in the solution may be limited and meshless weak solutions are favourable.

*Corresponding author: S. J. Lind, steven.lind@manchester.ac.uk, Tel.: +44 161 275 4260

Preprint submitted to Elsevier September 1, 2016
If greater regularity is imposed on particle distribution in ESPH and particles are fixed to a regularised grid, then equivalences can also be drawn to finite difference or highly accurate spectral methods where one uses collocation with Lagrange interpolants [44]. Indeed, the motivation for investigating ESPH and fixed particles is the possibility for solutions at higher orders of (spatial) accuracy (above second order). Some of the results presented herein will demonstrate levels of accuracy in the solutions that rival high-order compact/WENO finite-difference implementations [49, 67] and approach spectral methods [22] (on periodic domains). Issues around accuracy and convergence continue to befall standard SPH, which is often stated as being second order [46]. While this is true for standard symmetric normalised kernels obeying conditions on positivity with uniform particle spacing, advecting particles with the flow then introduces additional discretisation error that readily disrupts ideal second order accuracy [51]. To increase accuracy in Lagrangian SPH, a number of techniques have been employed. Normalisation approaches (e.g. [58, 52, 15]) and kernel gradient corrections (e.g. [11]) are able to improve accuracy for irregular spacing by enforcing zeroth-order (constant) and first-order (linear) consistency. Corrections of higher order accuracy can be derived that are also consistent [40], but these involve solving a set of equations for each particle and are likely to be costly at higher orders and in 3D, although very recent work has investigated explicit versions of such corrections [36]. Moving least squares approaches for hydrodynamics, originally due to Dilts [20], minimise a least squares functional to reconstruct (potentially high-order) polynomials based on surrounding particle values. The approach is once again implicit (and likely to be costly at higher orders and dimensions), but notable improvements in accuracy have been demonstrated with this approach in weakly compressible SPH when combined with a WENO scheme and an appropriate Riemann solver [3]. Particle regularisation procedures, such as shifting at each time step [66, 38] and particle re-meshing [14, 17] help by improving particle distribution and thereby reduce error due to non-uniformity, but perfect convergence is not recovered (when it has been considered). For example, studies often show observed convergence rates in the primitive variables lying between 1.5 and (less than) 2 [38]. A review of particle methods with some focus on re-meshing (and over multiple length scales) is given in [31]. Very recent work by Litvinov et al. [39] undertakes an iterative particle regularisation procedure (akin to shifting) that is able to recover particle distributions that show the ideal convergence in the discretisation error. However, such an approach is likely to be costly for transient simulations as iterative regularisation will need to be undertaken at each time step. As an alternative to the aforementioned techniques, this paper will demonstrate that Eulerian SPH with high-order kernels can offer straightforward improvements in convergence and accuracy quite efficiently for internal flows. Importantly, use of ESPH does not mean the attractive features of Lagrangian SPH are lost. It shall also be shown that the fully Lagrangian form can be coupled in a straightforward way where necessary (e.g. near free surfaces).

This paper presents a range of test cases which focus on transient and steady internal flows with and without solid boundaries. These include Taylor-Green vortices, Taylor-Couette flow, the lid-driven cavity, and confined channel flow around a cylinder. New high-order Gaussian kernels are derived and used to explore the amenability of ESPH to high-order solutions for the above cases. The versatility and potential of the method is then demonstrated through a free-surface flow test case (periodic wave propagation) where the Eulerian flow region merges to the standard Lagrangian treatment near the free surface, in a natural and straightforward manner. This is the first presentation of what can be termed an Arbitrary-Eulerian-Lagrangian (ALE) ISPH method. The manuscript is structured as follows: Section 2 outlines the SPH method, error characteristics, and presents the new higher-order smoothing kernels used. Section 3 discusses the gradient and Laplacian operators, while Section 4 presents the governing equations and the second order temporal discretisation used to solve them (in the transient flow case). Section 5 presents numerical results for the chosen test cases including discussions around convergence, accuracy and CPU time. Conclusions are given in Section 6.
2. SPH Interpolation and High-Order Kernels

In SPH, a variable $A$ at a point $r_i$ is approximated by a convolution product of the variable $A$ with a smoothing kernel function $\omega_h(|r_i - r|)$, with a smoothing length $h$, and is written as

$$A(r_i) \approx \int_{\Omega} A(r) \omega_h(|r_i - r|) dV,$$

(1)

where $\Omega$ is the supporting domain. In a discretised format, the interpolation can be written as

$$A(r_i) \approx \sum_j V_j A(r_j) \omega_h(r_{ij}),$$

(2)

where $V_j$ is the particle volume, and $r_{ij}$ the distance between particle $i$ and $j$. Hereafter $\omega_h(r_{ij})$ will be simply written as $\omega_{ij}$ or $\omega$ in the discrete and continuous cases, respectively. In addition to the requirement that $\omega$ tends to the Dirac delta function as $h \to 0$, standard SPH usually advocates the following two conditions on the smoothing kernel [41]:

1. Normalisation:

$$\int_{\Omega} \omega dV = 1$$

(3)

2. Positivity:

$$\omega \geq 0 \text{ in } \Omega$$

(4)

To introduce higher order approximations to (1), consider a Taylor series expansion about some point of interest $r_i = (x_i, y_i)$. Letting $r$ parameterise the line segment connecting $(x_i, y_i)$ with neighbouring points, we have

$$A(r) = A_i + A'_i r + \frac{1}{2} A''_i r^2 + \frac{1}{6} A'''_i r^3 + O(r^4)$$

(5)

where $A'_i = e_r \cdot \nabla A_i$ denotes the directional derivative along $r$ at point $i$, with $e_r$ the unit direction vector. Substituting (5) into (1) and non-dimensionalising kernel length scales with respect to the characteristic smoothing length, $h$, yields

$$A_i = A_i \int \omega dV + h \int A'_i s \omega dV + \frac{h^2}{2} \int A''_i s^2 \omega dV + \frac{h^3}{6} \int A'''_i s^3 \omega dV + O(h^4),$$

(6)

where $s = r/h$. If condition 1 is obeyed and the kernel is normalised then the first term in expansion (6) is identically $A_i$. If the kernel is symmetric then all terms where $\omega$ is multiplied by an odd power of $s$ are zero as the integrand is an odd function with an integral of zero over a radially symmetric domain. Consequently the error associated with smoothing/interpolation is $O(h^2)$. Removal of condition 2 (positivity) allows one to attain arbitrary (but even) orders of accuracy in smoothing error by constructing extended kernels that satisfy the higher order terms (or moments) in the Taylor expansion. As indicated by [46], removal of the $3^{rd}$ term on the RHS of (6) can be achieved with a kernel of the form:

$$\omega_4 = (A + B s^2) \omega,$$

(7)

where the subscript 4 denotes a kernel with a smoothing error that is now $O(h^4)$. Indeed, the argument can be generalised such that all even moments up to $2n$ vanish with a kernel of the form:

$$\omega_{2n} = \sum_{m=0}^{n-1} (A_m s^{2m}) \omega,$$

(8)

for $n = 1, 2, \ldots$. The constants $A_m$ can be determined through (6) and normalisation. Whilst the smoothing error with (8) may be $O(h^{2n})$, there is a discretisation (or quadrature) error associated with the approximation in Equation (2). This approximation is essentially the “midpoint rule” for integration,
which is effectively indistinct from the trapezoid rule when taken over a radius of infinite support with an
appropriately decaying kernel. Studies as early as Poisson in the 1820s noted the extraordinarily rapid
rate of convergence of the trapezoid rule with certain functions (Poisson observed geometric convergence
with elliptic integrals) [61]. The Gaussian is one such analytic function with suitably rapid and smooth
decay at infinity to demonstrate exponential convergence with the trapezoid rule [61]. In other words, application of the
Gaussian kernel (on a regular particle distribution) results in a discretisation error that is small compared
to the smoothing error. For kernels with compact support and different decay properties, Quinlan et al.
[51] showed that the discretisation error depends critically on the ratio of particle spacing to smoothing
length and on the boundary smoothness, \( \hat{\beta} \), of the kernel (\( \hat{\beta} \) being the largest integer such that the \( \hat{\beta} \)th
derivative and all lower derivatives are zero at the edge of the kernel support). The Gaussian is infinitely
smooth, while popular kernels such as the cubic and quintic spline have \( \hat{\beta} = 2 \) and \( \hat{\beta} = 4 \), respectively.
Compact kernels can then recover the ideal smoothing error, provided that the boundary smoothness is
sufficient and \( h \) is adjusted appropriately with particle spacing. Practically then, provided the particles
remain uniform and the first order error due to non-uniformity is removed [51], ideal convergence can
be achieved at higher-order. With the aforementioned discussion in mind, the Gaussian is chosen as our
“base kernel” from which we construct higher-order counterparts to use in simulations. Removal of the
2nd moment from (6) results in the 4th order Gaussian (G4) in 2D:

\[
\omega_4 = \left( \frac{2}{\pi h^2} \right) \left( 1 - \frac{r^2}{2h^2} \right) \exp\left( -\frac{r^2}{h^2} \right).
\]

Similarly, removal of the 4th moment gives a two-dimensional 6th order Gaussian kernel (G6):

\[
\omega_6 = \left( \frac{3}{\pi h^2} \right) \left( 1 - \frac{r^2}{h^2} + \frac{r^4}{6h^4} \right) \exp\left( -\frac{r^2}{h^2} \right).
\]

One could continue to construct higher-orders should one wish, but it will be demonstrated that the G6
is quite sufficient for attaining high accuracies at moderate particle spacing.

3. SPH Gradient and Laplacian Operators

3.1. The Gradient

SPH approximates a set of governing partial differential equations through substitution of appropriate
gradient (and, if required, Laplacian) operators. Two popular gradient approximations in SPH are:

\[
\nabla A_i \approx \pm \sum_j (A_i \pm A_j)V_j \omega_{ij}.
\]

In standard SPH, the negatively signed version maintains zeroth order consistency as the gradient of
any constant function \( A \) is zero. Meanwhile, the positive version is symmetric in the particle index and
thereby conserves particle-particle momentum. Consequently, the positive version of (11) is favoured by
those who wish to conserve properties associated with the discrete particle system over the immediate
 gains in accuracy at the continuum level that arise from the negative version. As ESPH treats particles
as fixed interpolation points the choice of sign in (11) becomes largely irrelevant, and both versions have
similar errors for uniform particle spacing [23]. To align with our previous work, we take the negative
version of (11). For a kernel of order \( O(h^{2n}) \), the dominant error term (and so convergence properties)
of the gradient approximation can be determined through a Taylor expansion and integration by parts
to give:

\[
E_{\nabla} = \frac{h^{2n}}{6} \sum \frac{A''_i s^3 V_j}{6} \omega_{ij}.
\]

Importantly, the theoretical convergence rate remains \( O(h^{2n}) \), and this will be demonstrated in due course
with numerical experiments.
3.2. The Laplacian

The Morris Laplacian [47] which combines a finite-difference approximation with the first derivative of the kernel has been popular due to its stability and accuracy for a range of Newtonian flow problems (e.g. 38). It is claimed by Basa et al. [4] to be one of the best Laplacian approximations available in SPH with Poiseuille flow Lagrangian particle tests reporting errors in velocity consistently less than 1%. Some alternative SPH approximations, such as those due to Flebbe et al. [26] or Violeau and Issa [64], yield errors between 1% and 5%, even if kernel corrections are employed [4]. In the context of particle methods generally, Degond and Mas-Gallic [19] provide important statements on convergence and stability with error bounds for integral approximations to the Laplacian in convection-diffusion problems. Their analysis holds for more general kernels, but imposition of a spherically symmetric kernel function effectively yields the approximation due to Morris et al. [47]. The generality of the treatment in [19] has been particularly attractive in the development of vortex particle methods (e.g. [65, 25]), where special cases of the Laplacian integral approximation [19] have been applied to a range of viscous flow problems (e.g. flows around impulsively started plates [32] and rotating cylinders [59]). The above discussion supports the adoption of the Morris formulation in this work; the formulation is a special case of the proven convergent integral Laplacian approximation in [19] and has been shown to be accurate in an SPH context. Accordingly, the Morris Laplacian reads

\[
\nabla^2 A_i = 2 \sum_j \frac{A_i - A_j}{r_{ij}} \nabla \omega_{ij} V_j, \tag{13}
\]

where \( \mathbf{e}_{ij} \) is the unit direction vector between points \( i \) and \( j \). In a similar argument to that used for the gradient, it can be shown that the lead smoothing error for the Morris Laplacian, \( E_{\nabla^2} \), is of equal order to that of the SPH interpolation and gradient. The Morris Laplacian is, therefore, suitable for high-order calculations. In particular, for a kernel of order 2n:

\[
E_{\nabla^2} = -\frac{2h^{2n}}{(2n+2)!} \sum A^{(2n+2)}_n s^{2n+1} \omega_{ij}' V_j, \tag{14}
\]

Despite remaining \( O(h^{2n}) \), compared with (12), the Laplacian error depends on higher derivatives of function \( A \). It is a natural requirement that, in the extension to higher orders, a greater degree of continuity is assumed in the solution, which may be disrupted at boundaries of the computational domain, at the boundaries of compactly supported kernels, or the truncated Gaussian domain. We expect, therefore, the Laplacian to be more prone to error, in general, than the first derivative.

3.3. Numerical Experiments Demonstrating Convergence in \( \nabla \) and \( \nabla^2 \)

The aforementioned error estimates will now be demonstrated numerically. Consider a periodic domain \([-0.5, 0.5] \times [-0.5, 0.5]\) containing a fixed uniform array of particles of spacing \( dx \). For a function, \( f \), over this domain given by

\[
f(x, y) = (\cos(4\pi x) + \cos(4\pi y)), \tag{15}
\]

the \( L_1 \) error in the gradient and the Laplacian of \( f \) will be studied. Note, we will take the horizontal component of the gradient to calculate the error, but the vertical component error magnitude and behaviour is identical. Figure 1 shows convergence in the error of \( \mathbf{e}_x \cdot \nabla f \) for the 2nd order (G2), 4th order (G4) and 6th order (G6) Gaussian-based kernels. All three kernels demonstrate near optimum convergence. Note that, as we are using Gaussian-based kernels (with associated exponential convergence in the discretisation error), the ratio of \( h \) to \( dx \) was fixed at \( h = 2dx \). Furthermore, to ensure that any error due to the truncation of the infinite Gaussian support domain is negligible, we select a large effective support radius of \( 6h \). For example, at the support boundary G2 has a value below machine precision. This is a large support domain that, of course, increases the number of neighbouring particles in the computation. For simulations using Lagrangian SPH this would be very expensive as neighbour searches and kernel calculations have to be undertaken anew every time step. This is not the case with ESPH. As particles are fixed, particle linking and kernel gradient calculations need only be executed once at the start of
the simulation. Therefore, despite a large support radius, the ESPH computation can be as fast as its
standard Lagrangian counterpart per time step (for \( dx = 0.025 \), tests showed CPU time per time step to
be approximately 0.123s and 0.139s for Eulerian SPH (with Gaussian kernel) and Lagrangian SPH (with
cubic spline kernel of support 3h), respectively.

Figure 2 shows the error convergence in \( \nabla^2 f \) for G2, G4, G6. Once again, near perfect smoothing error
is recovered in all three cases. Only in the case of G6 does the computation deviate slightly from the
theoretical order, and only when error reaches below \( 10^{-9} \). A similar, but smaller, deviation could be
seen in Fig. 1. Evidently, quadrature errors due to kernel truncation and discretisation of the smoothing
integral begin to materialise at this degree of accuracy. However, for practical computation, this slight
deviation from optimum is little cause for concern.

4. The Navier-Stokes Equations and Temporal Discretisation for Transient Flows

Consider the non-dimensional unsteady Navier-Stokes equations in Eulerian form:

\[
\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{u}. 
\]  

(16)

Of course, as the computation points are now fixed in space, the advection term is made explicit and
calculated at each particle using standard SPH techniques (Eqn. (11)). The Reynolds number \( Re \) is
defined in the usual way,

\[
Re = \frac{\rho U_m D}{\mu},
\]  

(17)

where \( \rho \) and \( \mu \) are the fluid density and viscosity, respectively. \( U_m \) and \( D \) are the characteristic flow
velocity and length scale, where \( U_m \) is usually the maximum velocity based on initial conditions. The
most popular means of integrating (16) in time is through the splitting of the primitive variables using
a projection method. First introduced by Chorin [16], the projection method was included in SPH by
[18] and further developed by the authors [66, 38] and others [37] for a range of flow problems. A key
advantage of using a projection-based incompressible SPH method over weakly-compressible SPH is that the resulting pressure field is smooth and non-oscillatory, provided the particle distribution is reasonably regular [38]. ESPH, therefore, has the capacity to attain highly accurate pressure fields for truly incompressible flows.

The projection method first introduced by [16] and used subsequently in SPH computations is formally first order accurate for velocity, and \( O(\sqrt{\Delta t}) \) for the pressure [30]. Therefore, for transient flows, to make the most of the increased accuracy through higher order kernels, a variant of the higher order projection method of Timmermans et al. [60] is applied. Guermond and Shen [28] showed that velocity is \( O(\Delta t^2) \) while pressure is formally \( O(\Delta t^{3/2}) \) with this scheme, but can reach \( O(\Delta t^2) \) in smooth domains (this will be demonstrated in Section 5). Slightly higher-order methods do exist (e.g. Heinrichs [29] proposes a 3rd order method) but issues exist over long-term stability [30]. According to [60], the first projection step utilises second order backwards differentiation to discretise the time derivative in (16) to give

\[
\frac{3\mathbf{u}^* - 4\mathbf{u}^n + \mathbf{u}^{n-1}}{2\Delta t} = -\nabla p^n + \frac{1}{Re} \nabla^2 \mathbf{u}^* + f^{n+1}.
\]  

(18)

Estimations of the advection term contained in \( f \) at time step \( n + 1 \) can be found through a second order extrapolation, e.g.

\[
f^{n+1} = 2f^n - f^{n-1} + O(\Delta t^2).
\]  

(19)

The divergence correction of \( \mathbf{u}^* \) is undertaken in the second step:

\[
\frac{3\mathbf{u}^{n+1} - 3\mathbf{u}^*}{2\Delta t} = -\nabla \left( (\delta p)^{n+1} + \frac{1}{Re} \nabla \cdot \mathbf{u}^* \right)
\]  

(20)

where \( \nabla \cdot \mathbf{u}^{n+1} = 0 \) and \( \delta p^{n+1} = p^{n+1} - p^n \). Note that combining (18) with (20) yields an appropriate temporal discretisation of (16). Defining \( q = (\delta p)^{n+1} + \frac{1}{Re} \nabla \cdot \mathbf{u}^* \), the following Poisson equation can be solved for \( q^{n+1} \).
∇^2 q^{n+1} = \frac{3}{2\Delta t} \nabla \cdot \mathbf{u}^*, \quad (21)

with a consistent Neumann condition $\partial q/\partial n = 0$ on the boundary of the flow domain. Upon solution of (21), the pressure at time $n + 1$ can be found from $p^{n+1} = q^{n+1} + p^n - \frac{1}{Re} \nabla \cdot \mathbf{u}^*$. The above pressure correction scheme has the advantage that for no-penetration boundaries, boundary conditions for the pressure are imposed correctly (i.e. they are consistent with the momentum equation (16)). Indeed, the inability to enforce appropriate boundary conditions for the pressure essentially limited the temporal accuracy of previous methods to first order [30]. As presented, the intermediate velocity $\mathbf{u}^*$ in (18) is treated implicitly given its presence in the diffusion term. To avoid the expense of solving an additional linear system and to be consistent with our previous work, we opt for an explicit velocity implementation of (18) where $\nabla^2 \mathbf{u}^*$ is approximated through extrapolation (19) to reside at time-step $n + 1$. By adopting an explicit projection scheme (for velocity), there are appropriate stability restrictions on the time step. In particular, the following dimensionless inertial and viscous constraints apply [50]:

$$\Delta t \leq \frac{C_l}{Re},$$

and

$$\Delta t \leq \frac{C_\mu}{Redx^2},$$

In the finite difference study of [24], the values of constants $C_l$ and $C_\mu$ are taken to be 2 and 1/6, respectively. Here we adopt conservative estimates and choose $C_l = 1$ and $C_\mu = 0.1$ for most cases. Note that Eqn. (22) is not the standard CFL condition, but the slightly stricter criterion derived from discretisations of the Navier-Stokes which include the hyperbolic advection term without special treatment [54]. At the start of each simulation, the smaller of the two time step constraints is taken.

In summary, utilising the discrete SPH operators of Section 3, the fully discrete system of governing equations reads:

- **The 1st step**

$$\frac{3\mathbf{u}^*_i - 4\mathbf{u}^n_i + \mathbf{u}^{n-1}_i}{2\Delta t} = - \sum_j V_j (p^*_j - p^n_j) \nabla \omega_{ij} + \left( \frac{1}{Re} \right) \sum_j V_j \frac{2r_{ij} \cdot \nabla \omega_{ij}}{r^2_{ij}} \mathbf{u}^{n+1}_{ij} + f^{n+1}_i. \quad (24)$$

- **The Poisson Equation**

$$\sum_j 2V_j \left( \frac{q^{n+1}_i - q^{n+1}_j}{r^2_{ij}} \mathbf{r}_{ij} \cdot \nabla \omega_{ij} \right) = \frac{3}{2\Delta t} \sum_j V_j (\mathbf{u}^*_j - \mathbf{u}^*_i) \cdot \nabla \omega_{ij}. \quad (25)$$

- **The 2nd (velocity correction) step**

$$\frac{3\mathbf{u}^{n+1}_i - 3\mathbf{u}^*_i}{2\Delta t} = - \sum_j V_j (q^{n+1}_j - q^{n+1}_i) \nabla \omega_{ij}.$$  

Equation (25) produces a linear equation system for $q^{n+1}$ that is solved iteratively using a stabilised biconjugate gradient method [63], as in [38, 66]. Of course, iterative methods find approximate solutions to linear systems where the accuracy is controlled by some preset tolerance. Here the tolerance setting used depends on the test case, but its effect on error and convergence will be discussed in due course.
5. Numerical Results

5.1. Taylor-Green Vortices

Consider the 2D Taylor-Green flow of a viscous incompressible fluid in a periodic domain with dimensions $[-0.5, 0.5] \times [-0.5, 0.5]$. The analytical expressions for primitive variables $p$, $u$, and $v$ are

\begin{align*}
    p &= e^{2bt}(\cos(4\pi x) + \cos(4\pi y)), \\
    u &= -e^{bt}\cos(2\pi x)\sin(2\pi y), \\
    v &= e^{bt}\sin(2\pi x)\cos(2\pi y),
\end{align*}

where $b = -\frac{8\pi^2}{Re}$. The flow consists of counter-rotating vortices decaying temporally due to viscosity; Figure 3 shows a typical pressure contour plot with streamlines over a fixed Cartesian particle distribution. All results in this subsection use a support radius of $6h$ with $h = 2dx$.

5.1.1. Accuracy Test for One Time Step

To initially test the accuracy of the aforementioned combination of gradient and Laplace operators (without the influence of the iterative solver), the numerical horizontal velocity is calculated at time-step $n = 1$ using a simple Euler approximation to the momentum equations (16) and analytical expressions for $p$, $u$, $v$ at $n = 0$. The time is chosen to be very small so as to minimise any truncation error from time integration. Figure 4 plots the $L_1$ error convergence in the velocity for each of the studied kernels. Both G2 and G4 demonstrate optimum convergence while G6 begins to converge optimally before being limited by machine precision. This demonstrates the potential gains in accuracy with high-order kernels and controlled particle distributions. Evidently, the more practical simulations that follow will be limited by time integration errors and the tolerance of the iterative solver. Note that to resolve non-uniqueness in the solution of (25) (an elliptic equation with purely homogenous Neumann boundary conditions), a Dirichlet condition is prescribed to a single corner particle using the analytical solution above.
5.1.2. Full Numerical Solution

Consider a low viscosity case, $Re = 1000$, where the temporal viscous decay in the solution is quite slow. The $L_1$ error in the numerical solution is measured at two sample times: $t = 0.1$ and 10. Figure 5 shows the convergence of the $L_1$ error in horizontal velocity, $u$, with particle spacing for the different Gaussian kernels (measured at $t = 0.1$). Note that we omit results for the vertical velocity, $v$, due to their similarity with those of $u$. Both G2 and G4 demonstrate optimum convergence, with accuracy as high as $3.3 \times 10^{-10}$ for $dx = 0.003125$ (G4). The 6th order kernel begins to converge well but plateaus at around $10^{-11}$. Almost identical behaviour is observed in the $L_1$ error in the pressure (see Fig. 6). The observed limit at 6th order is due to time integration error (in the case of velocity) and error introduced by the iterative solver (in the case of pressure). For example, halving the time step results in a reduction in velocity consistent with the second order time integration scheme (see Table 1). However, the error in the pressure remains limited by the iterative solver. Only when the tolerance of the solver is reduced from $1 \times 10^{-11}$ to $1 \times 10^{-12}$ does the pressure error reduce accordingly. While improvements to the pressure error can be achieved through decreases in tolerance, for the purpose of practical computation such small tolerances can be quite expensive. For the cases studied, while a tolerance of $1 \times 10^{-11}$ can take less than 100 iterations, $1 \times 10^{-12}$ can require between 500-1000. Application and preference will determine whether one wishes to recover 6th order convergence and accuracies (beyond $O(10^{-8})$ here) in the pressure, given the increased computational expense. For further reductions in time step, Table 2 demonstrates the continued second order convergence in velocity. While the pressure demonstrates similar second order convergence in the early stages (as predicted), it is once again limited by the solver tolerance at smaller time steps (Table 2). The velocity data from Table 2 is plotted in Figure 7, which shows how reductions in time step begin to recover the ideal 6th order spatial convergence. Recovery of ideal 6th order convergence is possible with appropriate reductions in tolerance and time step for this case, but, in practical simulations, given the exceptional accuracies already provided ($O(10^{-12})$), there is arguably little to gain given the additional computational expense.

Considering the longer-term behaviour of the method, at $t = 10$ the convergence properties and accuracy remain very good. Tables 3, 4 and 5 show $L_1$ error in velocity and pressure for a selection of particle spacings and the different kernels. Table 3 shows results from G2 with ideal 2nd order convergence being demonstrated in both $u$ and $p$. Table 4 shows errors for the 4th order Gaussian G4: ideal 4th order
Figure 5: $L_1$ error convergence in the horizontal velocity for 2nd order (circles), 4th order (triangles), and 6th order (squares) Gaussian-based kernel functions. The straight lines denote the theoretical ideal convergence. The measurement is taken at time $t = 0.1$.

Figure 6: $L_1$ error convergence in the pressure for 2nd order (circles), 4th order (triangles), and 6th order (squares) Gaussian-based kernel functions. The straight lines denote the theoretical ideal convergence. The measurement is taken at time $t = 0.1$. 
convergence is seen in the velocity, while error order for the pressure is near ideal initially before decreasing slightly as the error limits due to time stepping and solver tolerance are approached. For G6 (Table 5), the accuracy of the spatial approximation is such that the lower bound on error due to time integration error and tolerance is reached quickly. Convergence in velocity is near ideal initially before decreasing to around 4 between $dx = 0.0125$ and $dx = 6.25 \times 10^{-3}$. The error in the pressure reaches its limit almost instantly, with errors restricted to $O(10^{-6})$. Even at $t = 10$, in the presence of accrued time integration/solver errors, the G6 kernel provides results that are between 100 and 1000 times more accurate than the standard 2nd order Gaussian for a given particle spacing. 

As expected, similar convergence behaviour and gains in accuracy are observed with varying Reynolds number. For example, Table 6 shows the $L_1$ error in velocity and pressure for the three Gaussian kernels (G2, G4, G6) at $Re = 10$ and $t = 0.1$. Note that for G6, the pressure error is similarly limited by solver tolerance. For low Reynolds number ($Re < 1$) simulations, Equation (23) provides quite an increasingly severe restriction on time step size and, as such, these very viscous flows have not been studied. An implicit treatment of (24) would alleviate this issue, but, in any case, our primary interest is in flows of moderate to high Reynolds number where an explicit scheme remains practical.

Table 7 compares ESPH (with a 6th order kernel) with high-order finite difference based implementations [67, 49] for the Taylor-Green problem. Here the Reynolds number is $Re = 100$ and the $L_\infty$ norm is used as the measure for the absolute error in the horizontal velocity. Results are presented at two different times, $t = 1$ and $t = 10$. ESPH compares well to the other high-order implementations, and provides good accuracy over the longer-term. Note there are limited results available in the literature that show velocity errors at this Reynolds number, measured at these times and with this error norm, but the number of computation points used is comparable: ESPH has a resolution of $40 \times 40$ particles, Pandit et al. [49] utilise $40 \times 40$ cells on a non-uniform mesh, while the computations of Zhang and Jackson [67]
Figure 7: $L_1$ error convergence in the horizontal velocity for different time step sizes. The solver tolerance here is $1 \times 10^{-12}$.

| $dx$    | $u\ L_1$ error | Order | $p\ L_1$ error | Order |
|---------|-----------------|-------|-----------------|-------|
| 0.025   | $5.77 \times 10^{-5}$ | -     | $4.10 \times 10^{-4}$ | -     |
| 0.0125  | $3.67 \times 10^{-6}$ | 3.97  | $2.80 \times 10^{-5}$ | 3.87  |
| 0.00625 | $2.31 \times 10^{-7}$ | 3.99  | $3.99 \times 10^{-6}$ | 2.81  |

Table 4: $L_1$ Error in horizontal velocity and pressure at $t = 10$ for different particle spacings with associated orders of convergence. Results are obtained with 4th order Gaussian, G4.

are done on a regular $32 \times 32$ mesh. For a similar problem (a variant of Taylor-Green vortices) the spectral discretisation employed by Dumon et al. [22] reports accuracies of $1 \times 10^{-4}$ at $t = 1$ for polynomials of order $N = 16$. Some caution should be exercised in the interpretation of these comparisons as strict quantitative comparison is, of course, not possible. For example, not all the aforementioned schemes are formally of the same order of accuracy, and it is unknown if the tolerances used in the solutions of the associated linear systems (e.g. multi-grid [67] and BiCGStab [49]) are as stringent as used here. Nevertheless, the comparisons indicate, in as quantitative a manner as possible, that ESPH has the potential to compete with other highly-accurate numerical methods, while retaining the key meshless features of interpolative particle methods.

5.2. Steady Taylor-Couette Flow

While an important test of spatial and temporal accuracy, the Taylor-Green vortex case is somewhat idealised in having only periodic boundaries. To demonstrate the applicability of ESPH, subsequent test cases will include basic rigid boundaries and other standard SPH numerical settings to assess ESPH in a more practical context. In particular, a moderate solver tolerance of $1 \times 10^{-5}$ is now used within a first order projection scheme [38] that provides the iteration mechanism from initially unphysical flows to the physical steady state. The kernel support length is also reduced to the recommended $3h$ in the literature [46], with $h = 2\ dx$. The first such case to be investigated is steady Taylor-Couette flow which consists of viscous fluid confined between two concentric infinitely long cylinders, with the inner cylinder rotating with angular velocity $1\ \text{rads}^{-1}$ (see Figure 8). SPH particles are now small segments of an annulus and are distributed in a radial pattern with volumes $V_j = r_j\ d\theta_j dx$ chosen to recover exact partition of unity.
Table 5: $L_1$ Error in horizontal velocity and pressure at $t = 10$ for different particle spacings with associated orders of convergence. Results are obtained with 6th order Gaussian, G6.

| Kernel | $u L_1$ Error | Order | $p L_1$ Error | Order |
|--------|---------------|-------|---------------|-------|
| G2     | $2.2 \times 10^{-4}$ | - | $5.7 \times 10^{-4}$ | - |
| G4     | $2.4 \times 10^{-7}$ | 5.90 | $2.5 \times 10^{-6}$ | 4.19 |
| G6     | $9.4 \times 10^{-9}$ | 3.92 | $3.8 \times 10^{-7}$ | - |

Table 6: $L_1$ error in velocity and pressure for $Re = 10$ at $t = 0.1$ using kernels of different order. The particle spacing is $dx = 0.00625$.

One of the simplest SPH particle boundary conditions is implemented, so called dummy particles. Here a layer of particles are placed external to the domain to complete kernel support while being assigned the boundary velocity. Dummy particle densities are taken to match the fluid density which is constant everywhere. The imposition of accurate boundary conditions in SPH is an open problem, and a dedicated Grand Challenge posed by the SPH European Interest Community (SPHERIC). Nevertheless, considerable gains in accuracy using ESPH and high-order kernels shall be demonstrated even for these basic boundary conditions. The use of simple SPH boundary conditions here is important in demonstrating that ESPH remains straightforward to implement for practical problems while also providing gains in accuracy. Table 8 presents $L_1$ errors in the fluid velocity magnitude at steady state. The flow is assumed to have reached a steady state (from stationary initial conditions) when both horizontal and vertical velocity components satisfy the following condition:

$$\frac{\Delta w_{\text{max}}}{w_{\text{max}} \Delta t} < \epsilon,$$

where $\Delta w_{\text{max}}$ is the maximum difference in the velocity component between iterations over a maximum component value of $w_{\text{max}}$. The tolerance $\epsilon$ is chosen to be $1 \times 10^{-4}$ for this test case. Accordingly, from Eqn. (31), maximum relative changes in velocity components then have to be less than $10^{-6}$ for a steady state to be assumed, given the time step size used (initially $\Delta t = 10^{-2}$ here, chosen to satisfy the time step stability criteria, Eqns. (22) and (23)). Table 8 shows that the imposition of approximate (but practical, easily implementable) boundary conditions has destroyed the ideal convergence observed in the Taylor-Green test case; convergence is now close to linear in all cases. The errors obtained are several orders of magnitude larger than the largest relative change in velocity between iterations at the (approximate) steady state, indicating that error is dominated by the influence of boundary conditions rather than iteration to a steady state. Nevertheless, high order kernels are still able to provide up to order of magnitude decreases in the error with fractional increases in computational cost (at a fixed particle resolution).
| Scheme               | $u \ L_\infty$ Error at $t = 1$ | $u \ L_\infty$ Error at $t = 10$ |
|---------------------|--------------------------------|---------------------------------|
| WENO (5th order)    | $2.98 \times 10^{-6}$          | $2.12 \times 10^{-4}$          |
| 4th order compact   | $1.89 \times 10^{-3}$          | $1.60 \times 10^{-3}$          |
| 6th order compact   | $2.82 \times 10^{-6}$          | $8.59 \times 10^{-6}$          |
| ESPH (G6)           | $1.66 \times 10^{-5}$          | $1.35 \times 10^{-7}$          |

Table 7: Comparison of ESPH (using a 6th order Gaussian) with high-order finite-difference based schemes for Taylor-Green vortices for $Re = 100$.

With regard to increases in computational efficiency, Table 9 shows the CPU times required for an $L_1$ error of approximately $2.23 \times 10^{-2}$ to be achieved for each of the studied kernels. The simulations were run in serial on a 2.1 GHz Intel Xeon CPU with 64 GB of RAM. The second order (G2) kernel requires a particle resolution of $dx = 0.025$ to attain errors of $2.23 \times 10^{-2}$ and does so in 4hrs 5mins. The sixth order kernel can attain this accuracy using a resolution of $dx = 0.1$ in 2mins. For this case, G6 can thus provide accuracies equivalent to second order kernels but in less than 1% of the CPU time. This is a clear demonstration that if one is satisfied with the current level of accuracy when using second order kernels, higher orders can offer exceptional improvements in efficiency by allowing lower particle resolutions. In all tested simulations, the set-up and solution of the PPE linear system took between 80-90% of the CPU time, regardless of particle discretisation or kernel used. Calculations pertaining to the remaining gradient and Laplacian terms in the governing equations (e.g. the advection, viscous diffusion, and pressure gradient terms) took 2-3% of the CPU time each. Any remaining time was associated with allocation and deallocation of minor arrays, geometry input, initialisation, and results output.

To demonstrate that high order convergence remains achievable for this flow and non-Cartesian geometry, the analytical solution is now imposed everywhere initially to remove errors due to boundary conditions and iteration to steady state. Table 10 shows the $L_1$ error in the calculated velocity magnitude for the different kernels and particle resolutions after one iteration from steady state. Ideal spatial convergence is recovered throughout.

5.3. Lid-Driven Cavity

The lid-driven cavity remains a popular validation case for incompressible flows. The square $[0, 1] \times [0, 1]$ domain is shown in Figure 9, with the top boundary moving horizontally at speed $U$ and driving the internal flow to a steady state at low to moderate Reynolds numbers ($Re = 100$ and 400 are considered...
Table 8: $L_1$ error in velocity for Taylor-Couette flow with dummy particle boundary conditions. Orders of convergence are given column-wise in brackets.

| $dx$   | Second         | Fourth         | Sixth          |
|--------|----------------|----------------|----------------|
| 0.1    | $8.24 \times 10^{-2}$ (-) | $2.39 \times 10^{-2}$ (-) | $6.25 \times 10^{-3}$ (-) |
| 0.05   | $4.14 \times 10^{-2}$ (1.00) | $1.07 \times 10^{-2}$ (1.16) | $3.71 \times 10^{-3}$ (0.75) |
| 0.025  | $2.23 \times 10^{-2}$ (0.89) | $6.07 \times 10^{-3}$ (0.82) | $1.79 \times 10^{-3}$ (1.05) |

Table 9: CPU time and particle resolution required (from $dx \in \{0.1, 0.05, 0.025\}$) to attain $L_1$ velocity error of $2.23 \times 10^{-2}$ for different kernels.

| Kernel | CPU time    | Required $dx$ |
|--------|-------------|---------------|
| G2     | 4hrs 5mins  | 0.025         |
| G4     | 11mins      | 0.05          |
| G6     | 2mins       | 0.1           |

5.4. Steady Confined Channel Flow Around a Cylinder

Flow around a cylinder is a fundamental test case in fluid mechanics and a good test for viscous flow predictions given the wealth of data available in the literature. The flow around a confined cylinder is considered here, with a cylinder of radius $R = 1$ embedded centrally in a horizontal channel of height $4$ units (giving a blockage ratio $\beta = 2$). A schematic of the problem domain is shown in Figure 14. At the inlet boundary a Poiseuille flow velocity profile ($U = -\frac{1}{2}(y-2)(y+2)$) is imposed with zero pressure at the outlet. Results are considered for two different SPH boundary conditions on the cylinder and upper and lower walls: the standard dummy particle boundary condition (described in Section 5.2), and the Adami et al. [1] boundary condition. This condition is an extension of the dummy particle method, and improves accuracy while retaining the flexibility of dummy boundary particles. The key difference is that all boundary particles are assigned an opposing fluid velocity using a Shepard interpolation [58],
Figure 9: Schematic of lid-driven cavity flow domain.

Figure 10: Horizontal velocity along the vertical centreline of the lid-driven cavity calculated at $Re = 100$ with ESPH for G2, G4, and G6. Squares denote the benchmark multigrid results of [27].
Figure 11: Vertical velocity along the horizontal centreline of the lid-driven cavity calculated at $Re = 100$ with ESPH for G2, G4, and G6. Squares denote the benchmark multigrid results of [27].

Figure 12: Horizontal velocity along the vertical centreline of the lid-driven cavity calculated at $Re = 400$ with ESPH for G2, G4, and G6. Squares denote the benchmark multigrid results of [27].
Table 10: $L_1$ error in velocity for Taylor-Couette flow with exact initial and boundary conditions. Orders of convergence are given column-wise in brackets.

| $dx$  | Second     | Fourth     | Sixth      |
|-------|------------|------------|------------|
| 0.05  | $4.07 \times 10^{-3}$ (-) | $8.02 \times 10^{-3}$ (-) | $2.37 \times 10^{-6}$ (-) |
| 0.025 | $1.05 \times 10^{-3}$ (1.95) | $5.17 \times 10^{-6}$ (3.96) | $3.74 \times 10^{-8}$ (5.99) |
| 0.0125| $2.69 \times 10^{-4}$ (1.96) | $3.32 \times 10^{-7}$ (3.96) | $6.01 \times 10^{-10}$ (5.96) |

Figure 13: A close-up of Figure 12 highlighting the discrepancy present when using second order kernel, G2.

$$u_{bi} = -\frac{\sum_j u_j W_{ij}}{\sum_j W_{ij}},$$

rather than being given the actual boundary velocity (zero in this case). Here $u_{bi}$ is the velocity of boundary particle $i$, and the summation is over internal fluid particles, $j$, only. As in Adami et al. [1], the kernel, $W_{ij}$, used in Eqn. (32) is second order in smoothing length. By including interpolated boundary velocities from Eqn. (32) in the calculation of the viscous term in Eqn. (16), the no-slip condition can be better approximated. For this test case, the steady flow regime is considered (which occurs when $Re \lesssim 120$ for $\beta = 2$ [57]), allowing a smaller computational domain and reduced computation time than for unsteady vortex shedding. Numerical experiments by the authors suggest that an upstream and downstream length of $10R$ (giving a total domain length of $20R$) is sufficient for obtaining drag coefficients with near streamwise domain independence. Once again, fairly coarse particle resolutions are considered to demonstrate the gains in accuracy available with high-order kernels. An example illustration of the flow at $Re = 15$ is given in Figure 15 which shows a typical steady state solution at $dx = 0.05$ with longitudinal velocity contours (obtained using the Adami et al. [1] boundary condition). Table 11 shows drag coefficient values calculated at $Re = 15$ for different kernels and particle resolutions, for the two different boundary conditions. The drag coefficient, $C_d$, is defined as

$$C_d = \frac{F_d}{\rho U_m^2 R},$$

where $U_m$ is the maximum velocity at the inlet and
\[
F_d = e_x \cdot \int_C -p n + \mu (\nabla u + \nabla u^T) \cdot n \, ds,
\]
(34)

with integration over the surface of the cylinder with normal \( n \). The error measurements in Table 11 (in parentheses) are calculated using drag coefficient results from the highly resolved simulations of Bharti et al. [10], who use the commercial software package Fluent. Note that the Reynolds number and viscous drag coefficient are defined differently in [10], but exact comparisons are still possible: their Reynolds number is based on an average velocity and is exactly two-thirds the Reynolds number used here (Eqn. (17)) and in [57] (based on the maximum inlet velocity). Similarly, the viscous drag coefficient used in [10] is exactly twice that used here. Using standard dummy particles, ESPH results show that, once again, higher order kernels offer systematic improvement in drag coefficient predictions. Note that the most resolved simulations here are still quite coarse relative to [10], who utilise grid spacings around the cylinder of 0.01 units. Importantly (and similarly to Taylor-Couette flow), use of 6th order kernels at particle spacing \( dx \) achieves accuracies as good as 2nd order kernels with spacing \( dx/2 \). Results of equivalent accuracy can again be obtained more quickly by using higher order kernels with fewer particles. As for the Taylor-Couette flow, all errors converge approximately linearly in the presence of dummy particles. Adoption of the Adami et al. [1] boundary condition provides considerable increases in accuracy, with up to an order of magnitude decrease in error available. For G2, a second-order decrease in error is now observed with decreasing particle spacing, but the error soon becomes limited at around 0.5% for the high-order kernels. As discussed in Section 5.2, the tolerance is sufficiently small so as to suggest negligible iteration error at steady state, so it is likely that the limiting difference of 0.5-0.6% is due to the loss of exact partition of unity in the domain near the top and bottom walls. As in the Taylor-Couette flow case, a perfectly arranged set of annular segment particles span outwards from the cylinder, but these now cross horizontal planar walls at the top and bottom of the flow domain. Without adjusting near wall particle volumes appropriately, some particles will overlap the boundary and the imposition of no-slip conditions along the side walls cannot be as accurate as on the cylinder surface. This effect is illustrated in Fig. 16 which plots the horizontal velocity along the line \( x = 0 \) (through the centre of the cylinder) for the two different boundary treatments (dummy particles and Adami et al. [1]). Clearly, the no-slip condition on the cylinder is well maintained, but it holds only approximately at the horizontal walls. Nevertheless, there are notable improvements in adherence to the no-slip condition at the side walls when using the Adami et al. [1] boundary condition, in line with the improved results presented in Table 11.

To demonstrate that the gains in accuracy with high-order kernels remain for a range of Reynolds numbers (and even for dummy boundary particles), Figure 17 shows the drag coefficients for \( \beta = 2 \) calculated for several Reynolds numbers in the steady regime. ESPH predictions are plotted for the three Gaussian kernels and compared with the highly resolved finite volume results of [57]. The ESPH results are demonstrative and of a low resolution (\( dx = 0.1 \)), but, importantly, the increase in accuracy through the use of high-order kernels remains evident for all studied Reynolds numbers. Of course, as demonstrated in Table 11, these results can be improved (if required) with increasing particle resolution and improved boundary conditions.

| Boundary condition | \( dx \) | Second, G2 | Fourth, G4 | Sixth, G6 |
|--------------------|--------|------------|------------|-----------|
| Dummy              | 0.1    | 7.97 (23.1)| 8.89 (14.3)| 9.21 (11.1)|
|                    | 0.05   | 9.24 (10.9)| 9.75 (6.0 )| 9.82 (5.3 )|
| Adami et al. [1]   | 0.1    | 9.47 (8.7 )| 10.25 (1.2)| 10.43 (0.6)|
|                    | 0.05   | 10.13 (2.3)| 10.42 (0.5)| 10.43 (0.6)|

Table 11: Drag coefficients \( C_d \) calculated using ESPH with different boundary conditions, kernels, and particle resolutions at \( Re = 15 \). Error measurements (\% in parentheses) are calculated with respect to the \( C_d = 10.37 \) value obtained from the well-refined simulations of [10].
Figure 14: Schematic of domain for channel flow around a cylinder.

Figure 15: Typical steady state solution for channel flow around a cylinder at $Re = 15$, $dx = 0.05$, using the Adami et al. \[1\] boundary condition. Contours denote horizontal velocity values, $u$. 
Figure 16: Horizontal velocity plotted along the centreline $x = 0$ for channel flow around a cylinder ($Re = 15$, $dx = 0.05$) for two different boundary conditions (dummy particles and Adami et al. [1]).

Figure 17: Variation of $C_d$ with $Re$ for different kernels, including the finite volume results of Sahin and Owens [57]. ESPH uses a very coarse resolution of $dx = 0.1$ and the dummy particle boundary condition for demonstrative purposes.
5.5. Extensions to Free-Surface Flow: Periodic Wave Propagation and the Future

While Eulerian SPH with high order kernels offers increased accuracy and efficiency, the purely Eulerian form is not generally applicable to free-surface flows; the ease of modelling being a key benefit of traditional SPH over grid based methods. As mentioned, the attractive Lagrangian features of SPH can be coupled in a straightforward and natural way from ESPH by merging the Eulerian framework with the Lagrangian as one approaches a free surface (or similar material discontinuity). This approach has the potential to allow high accuracies/reduced resolutions in the fluid bulk, while accuracy at the free surface may be recovered through adaptive refinement (where a number of recent works have shown promising results, e.g. [62]).

Here a test case is presented showing the ease with which ESPH may transition to standard Lagrangian SPH. This case only demonstrates the functionality of this new Arbitrary-Lagrangian-Eulerian (ALE) ISPH formulation, with quantification of the gains in accuracy available through the use of high-order kernels and adaptive/local particle refinement requiring further detailed study. Therefore, in this first instance, the ALE-ISPH formulation is based on the accurate Lagrangian methodology of Lind et al. [38], where a first-order projection method is used with second order corrected kernels and shifting.

Consider a fully non-linear inviscid progressive gravity wave over a periodic domain of length equal to the wavelength $\lambda \approx 2$. The wave height is $H = 0.1$ and the still water depth $D = 0.5$, with the initial free surface profile and flow velocities determined from irrotational stream function theory [53] and highly accurate iterative solution methods [13]. Figure 18 shows the domain and initial wave profile with contours coloured according to horizontal velocity.

Slip boundary conditions are imposed at the bed ($y = 0$) and a normalised gravity body force is applied, $|\mathbf{g}| = 1$. The Reynolds number in Eqn. (35) is set to $Re = 1000$ and therefore the flow is not strictly inviscid nor irrotational, but this has little effect over the first wave period, $T = 3.7$. To control the merging from Eulerian to Lagrangian regions, a transition function $\alpha(x) \in [0, 1]$ is introduced into the dimensionless governing equations,

$$\frac{\partial \mathbf{u}}{\partial t} |_{X_{\alpha}} + (1 - \alpha) \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{u} + \mathbf{g},$$

(35)

where $\alpha = 1$ denotes a fully Lagrangian treatment (with $\alpha = 0$ for fully Eulerian). Accordingly, $\frac{\partial \mathbf{u}}{\partial t} |_{X_{\alpha}}$ is
the local time derivative with coordinates $\mathbf{X}_\alpha$ fixed in reference frame $\alpha$, so $\frac{D\mathbf{u}}{Dt} = \frac{2n}{\partial t} \mathbf{x}_1$. These equations are solved in the usual way with SPH particles updated at the end of each time step according to

$$\mathbf{x}^{n+1} \approx \mathbf{x}^n + \alpha \mathbf{u}^n \Delta t. \quad (36)$$

Note that stabilising Fickian shifting \cite{38} is turned on only when particles move and $\alpha > 0$. The transition function $\alpha$ is constructed using basic knowledge of the flow geometry (a periodic wave of depth $D = 0.5$ and wave height $H = 0.1$). A smooth variation from Eulerian to Lagrangian regions is chosen in this first instance with

$$\alpha(y) = \frac{1}{2} (\tanh(\gamma(y - D_E)) + 1), \quad (37)$$

where the transition steepness parameter $\gamma = 25$ and $D_E$ denotes the $y$-position of centre of the transition region (taken to be 0.3 distance units from the bed). There are clearly a range of permissible transition functions (that may be dynamic or adaptive depending on the flow), and further study on ALE-ISPH will investigate the influence of transition functions and parameters on flow accuracy. After running the simulation for one wave period from an initially regular particle distribution, Figure 19 shows the developed particle distribution around the transition region. Those particles towards the bottom of the figure (the Eulerian region) remain in a Cartesian-like arrangement, before smoothly evolving into a Lagrangian distribution typical of standard SPH. Furthermore, the wave profile and velocity contours at $t = 1T$ (Fig. 20) closely resemble the initial conditions (Fig. 18).

To further validate the ALE-ISPH approach, Figure 21 shows the horizontal velocity (Fig. 21(a)) and non-hydrostatic pressure (Fig. 21(b)) vertically below the crest and the trough measured at the first wave period. Comparisons are made with semi-analytical solutions from irrotational stream function theory and the SAWW code \cite{13}. The non-hydrostatic pressure, $p^*$, is that pressure remaining when the dominant hydrostatic part is removed. In dimensional form it is defined as,

$$p^* = p - \rho|\mathbf{g}|(\eta + D - y), \quad (38)$$
where η is the wave elevation at a chosen horizontal location. It is a particularly sensitive flow measure, but both $p^*$ and $u$ remain in good agreement with irrotational stream-function theory (Fig. 21), despite ALE-ISPH including small viscous effects. Table 12 quantifies the error observable in Fig. 21 by presenting an average relative deviation of ALE-ISPH results from the irrotational solution. For both $p^*$ and $u$, under either a crest or trough, the average deviation remains less than 10%.

Table 12: The average relative deviation of ALE-ISPH predictions for $p^*$ and $u$ from the irrotational solution. Under either the crest or trough, the relative deviation is calculated at equispaced positions separated by 0.05 distance units along the vertical before the mean is taken.

|        | $p^*$  | $u$    |
|--------|--------|--------|
| Trough | $3.3 \times 10^{-2}$ | $8.3 \times 10^{-2}$ |
| Crest  | $3.1 \times 10^{-2}$ | $5.0 \times 10^{-2}$ |

5.5.1. Future Developments

Since convergence and accuracy in SPH benefits from regular particle distributions the Eulerian formulation has been thoroughly investigated. High-order spatial convergence and very good levels of accuracy (or efficiency) have been demonstrated for fixed particles, while the capability of merging the Eulerian form with the attractive Lagrangian features of SPH has been shown using an inviscid free-surface flow test case. A detailed study into this new ALE-ISPH formulation will be undertaken for a range of free surface flow test cases, and will investigate the effect of the transition region and the use of high-order kernels combined with variable particle resolution. The ALE-ISPH approach demonstrated herein is likely to be of higher accuracy and more convenient than undertaking coupling of two different numerical methods (one mesh based, possibly Eulerian or semi-Lagrangian, which may have good efficiency for large domains, and the other Lagrangian SPH with good generality but high computational cost). Such couplings are increasingly popular but are sensitive to numerical interface errors, inconsistent mathematical formulations (e.g. coupling potential and viscous flows), and are troublesome to implement for parallel processing.
Even in a purely Eulerian form, the approach herein has a number of key features which may enable it to compete with more established methods for many applications in the near future. The interpolative nature of (E)SPH simply requires unordered summation over surrounding computation points; there is no consideration of ordered node labelling (local or global), node connectivity, element/cell mappings, shapes, orientations, or interface conditions. Accordingly, “mesh generation” in complex geometries has the potential to be greatly simplified, requiring only a suitable distribution of unconnected particles. Secondly, the simplicity of SPH (mathematically and in programming) greatly eases parallel implementations and extension to 3D, as the discrete governing equations are essentially a series of summations, with the fixed particles easily partitioned across parallel processes. Solid boundary conditions remain an issue, and can restrict ideal convergence (as they do for standard 2nd order SPH). Nevertheless, even for the basic (and easily implementable) dummy particle boundary conditions, there are order of magnitude gains in accuracy available when using high order kernels and fixed particles. There may be options available to recover higher order convergence with boundaries in practical situations in the near future: the unified semi-analytical boundary conditions [37] have performed well in standard ISPH and demonstrated near-ideal smoothing convergence at 2nd order. The compatibility of these boundary conditions with high-order ESPH remains to be investigated.

6. Conclusions

This paper has investigated a new Eulerian Incompressible Smoothed Particle Hydrodynamics formulation to high order. By adopting an Eulerian framework, particle distributions remain uniform and high accuracies and/or computational savings are available, as demonstrated for a range of transient and steady internal viscous flows. Over periodic domains, high accuracies and ideal high order spatial convergence can be achieved in both the velocity and pressure, rivaling high-order finite-difference and approaching spectral methods at similar resolutions. Gaussian-based kernels are the most appropriate (due to their rapid convergence in the discretisation error) and two new higher order versions (fourth and sixth order in h) are derived in two dimensions. The attractive Lagrangian features of SPH can be recovered straightforwardly near material interfaces, and an inviscid free-surface flow test case is used to demonstrate the viability
of a new Arbitrary-Lagrangian-Eulerian (ALE) ISPH approach. The potential of ALE-ISPH is discussed and future developments are outlined. Even in a purely Eulerian form, incompressible SPH has a number of beneficial features, including ease of implementation and extension to 3D, and amenability to parallel processing.

Acknowledgements

The support of EPSRC grants EP/L014661/1 and EP/L014890/1 is gratefully acknowledged.

7. References

[1] Adami, S., Hu, X., Adams, N., 2012. A generalized wall boundary condition for smoothed particle hydrodynamics. J. Comput. Phys. 231 (21), 7057 – 7075.
[2] Atluri, S. N., Zhu, T., 1998. A new meshless local Petrov-Galerkin (MLPG) approach in computational mechanics. Comput. Mech. 22 (2), 117–127.
[3] Avesani, D., Dumbser, M., Bellin, A., 2014. A new class of moving-least-squares WENO-SPH schemes. J. Comput. Phys. 270, 278 – 299.
[4] Basa, M., Quinlan, N., Lastiwka, M., 2009. Robustness and accuracy of SPH formulations for viscous flow. Int. J. Numer. Meth. Fluids 60, 1127–1148.
[5] Beissel, S., Belytschko, T., 1996. Nodal integration of the element-free Galerkin method. Comput. Methods Appl. Mech. Engrg. 139 (1), 49–74.
[6] Belytschko, T., Krongauz, Y., Organ, D., Fleming, M., Krysl, P., 1996. Meshless methods: an overview and recent developments. Comput. Methods Appl. Mech. Engrg. 139 (1), 3–47.
[7] Belytschko, T., Lu, Y., Gu, L., 1994. Element-free Galerkin methods. Int. J. Numer. Meth. Eng. 37 (2), 229–256.
[8] Belytschko, T., Lu, Y., Gu, L., 1995. Crack propagation by element-free Galerkin methods. Eng. Fract. Mech. 51 (2), 295–315.
[9] Belytschko, T., Lu, Y., Gu, L., Tabbara, M., 1995. Element-free Galerkin methods for static and dynamic fracture. Int. J. Solids Struct. 32 (17), 2547–2570.
[10] Bharti, R. P., Chhabra, R. P., Eswaran, V., 2007. Two-dimensional steady Poiseuille flow of power-law fluids across a circular cylinder in a plane confined channel: Wall effects and drag coefficients. Ind. & Eng. Chem. Res. 46 (11), 3820–3840.
[11] Bonet, J., Lok, T. S. L., 1998. Variational and momentum preservation aspects of smooth particle hydrodynamics formulations. Comput. Method Appl. Mech. Engrg. 43, 785–819.
[12] Botella, O., Peyret, R., 1998. Benchmark spectral results on the lid-driven cavity flow. Comput. Fluids 27 (4), 421–433.
[13] Buss, G. Y., Stansby, P. K., 1982. SAWW - a computer program to calculate the properties of steady water waves. Tech. rep., Simon Engineering Laboratories, University of Manchester.
[14] Chaniotis, A., Poulikakos, D., Koumoutsakos, P., 2002. Remeshed smoothed particle hydrodynamics for the simulation of viscous and heat conducting flows. J. Comput. Phys. 182, 67–90.
[15] Chen, J., Beraun, J., Carney, T., 1999. A corrective smoothed particle method for boundary value problems in heat conduction. Int. J. Numer. Methods Engrg. 46, 231–252.
[16] Chorin, A. J., 1968. Numerical solution of the Navier Stokes equations. J. Math. Comp. 22, 745–762.
[17] Cottet, G.-H., Weynans, L., 2006. Particle methods revisited: a class of high-order finite-difference schemes. Comptes Rendus de l Académie des Sciences-Series I-Mathematics 343, 51–56.
[18] Cummins, S. J., Rudman, M., 1999. An SPH projection method. J. Comput. Phys. 152, 584–607.
[19] Degond, P., Mas-Gallic, S., 1989. The weighted particle method for convection-diffusion equations. I. The case of an isotropic viscosity. Math. Comput. 53 (188), 485–507.
[20] Dilts, G., 1999. Moving-least-squares-particle hydrodynamics I. Consistency and stability. Int. J. Numer. Methods Eng. 44 (8), 1115–1155.
[21] Dolbow, J., Belytschko, T., 1999. Numerical integration of the Galerkin weak form in meshfree methods. Comput. Mech. 23 (3), 219–230.
[22] Dumon, A., Allery, C., Ammar, A., 2013. Proper generalized decomposition method for incompressible Navier-Stokes equations with a spectral discretization. Appl. Math. Comput. 219 (15), 8145–8162.
[23] Fatehi, R., Manzari, M., 2011. Error estimation in smoothed particle hydrodynamics and a new scheme for second derivatives. Comput. Math. Appl. 61, 482–498.
[24] Fernandez-Feria, R., Sanmiguel-Rojas, E., 2004. An explicit projection method for solving incompressible flows driven by a pressure difference. Comput. Fluids 33, 463–483.
[25] Fishelov, D., 1990. A new vortex scheme for viscous flows. J. Comput. Phys. 86 (1), 211–224.
[26] Flebbe, O., Munzel, S., Herold, H., Riffert, H., Ruder, H., 1994. Smoothed particle hydrodynamics: physical viscosity and the simulation of accretion disks. Astrophys. J. 431, 754–760.
[27] Ghia, U., Ghia, K., Shin, C., 1982. High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method. J. Comput. Phys. 48 (3), 387–411.
[28] Guermond, J. L., Shen, J., 2004. On the error estimates for the rotational pressure-correction projection methods. Math. Comp. 73, 1719–1737.
[29] Heinrichs, W., 1998. Splitting techniques for the unsteady Stokes equations. SIAM J. Numer. Anal. 35, 1646–1662.
[30] Karniadakis, G., Sherwin, S. J., 2005. Spectral/hp element methods for computational fluid dynamics. Oxford University Press.
[31] Koumoutsakos, P., 2005. Multiscale flow simulations using particles. Annu. Rev. Fluid Mech. 37, 457–487.
[32] Koumoutsakos, P., Shiels, D., 1996. Simulations of the viscous flow normal to an impulsively started and uniformly accelerated flat plate. J. Fluid Mech. 328, 177–227.
[33] Krongauz, Y., Belytschko, T., 1996. Enforcement of essential boundary conditions in meshless approximations using finite elements. Comput. Methods Appl. Mech. Engrg. 131 (1), 133–145.
[34] Krysl, P., Belytschko, T., 1996. Analysis of thin shells by the element-free Galerkin method. Int. J. Solids Struct. 33 (20), 3057–3080.
[35] Lancaster, P., Salkauskas, K., 1981. Surfaces generated by moving least squares methods. Math. Comput. 37 (155), 141–158.
[36] Leonardi, M., Rung, T., 2014. Explicit strategies for consistent kernel approximations. In: Proceedings of the 9th SPHERIC Workshop. Paris, pp. 111–119.

[37] Leroy, A., Violeau, D., Ferrand, M., Kassiotis, C., 2014. Unified semi-analytical wall boundary conditions applied to 2-D incompressible SPH. J. Comput. Phys. 261, 106 – 129.

[38] Lind, S., Xu, R., Stansby, P., Rogers, B., 2012. Incompressible smoothed particle hydrodynamics for free-surface flows: A generalised diffusion-based algorithm for stability and validations for impulsive flows and propagating waves. J. Comput. Phys. 231 (4), 1499 – 1523.

[39] Litvinov, S., Hu, X., Adams, N., 2014. Towards both numerical consistency and conservation for SPH approximation. In: Proceedings of the 9th SPHERIC workshop. Paris, pp. 283–289.

[40] Liu, M., Liu, G., 2006. Restoring particle consistency in smoothed particle hydrodynamics. Appl. Numer. Math. 56 (1), 19–36.

[41] Liu, M., Liu, G., 2010. Smoothed particle hydrodynamics (SPH): an overview and recent developments. Arch. Comput. Methods. Eng. 17, 25–76.

[42] Lucy, L., 1977. Numerical approach to the testing of the fission hypothesis. Astron. J. 82, 1013–1024.

[43] Melenk, J., Babuška, I., 1996. The partition of unity finite element method: Basic theory and applications. Comput. Methods Appl. Mech. Engrg. 139 (1-4), 289 – 314.

[44] Monaghan, J. J., 1982. Why particle methods work. SIAM J. Sci. Stat. Comp. 3 (4), 422–433.

[45] Monaghan, J. J., 1985. Particle methods for hydrodynamics. Computer Physics Reports 3, 71–124.

[46] Monaghan, J. J., 1992. Smoothed particle hydrodynamics. Annu. Rev. Astron. Astrophys. 30, 543–574.

[47] Morris, J. P., Fox, P. J., Zhu, Y., 1997. Modelling low Reynolds number incompressible flows using SPH. J. Comput. Phys. 136, 214–226.

[48] Noutcheuwa, R. K., Owens, R. G., 2012. A new incompressible smoothed particle hydrodynamics-immersed boundary method, Int. J. Numer. Anal. Mod. B 3, 126–167.

[49] Pandit, S., Kalita, J., Dalal, D., 2007. A transient higher order compact scheme for incompressible viscous flows on geometries beyond rectangular. J. Comput. Phys. 225, 1100–1124.

[50] Peyret, R., Taylor, T. A., 1983. Computational methods for fluid flow. Springer-Verlag.

[51] Quinlan, N. J., Basa, M., Lastiwka, M., 2006. Truncation error in mesh-free particle methods. Int. J. Numer. Meth. Engrg. 66, 2064–2085.

[52] Randles, P., Libersky, L., 1996. Smoothed particle hydrodynamics some recent improvements and applications. Comput. Methods Appl. Mech. Engrg. 138, 375–408.

[53] Rienecker, M. M., Fenton, J. D., 1981. A Fourier approximation method for steady water waves. J. Fluid Mech. 104, 119–137.

[54] Rigal, A., Aleix, G., 1978. Stability analysis of some finite difference schemes for the Navier-Stokes equations. Int. J. Numer. Methods Eng. 12 (9), 1399–1405.

[55] Sahin, M., Owens, R. G., 2003. A novel fully implicit finite volume method applied to the lid-driven cavity problem - Part I: High Reynolds number flow calculations. Int. J. Numer. Meth. Fluids 42 (1), 57–77.
[56] Sahin, M., Owens, R. G., 2003. A novel fully-implicit finite volume method applied to the lid-driven cavity problem - Part II: Linear stability analysis. Int. J. Numer. Meth. Fluids 42 (1), 79–88.

[57] Sahin, M., Owens, R. G., 2004. A numerical investigation of wall effects up to high blockage ratios on two-dimensional flow past a confined circular cylinder. Phys. Fluids 16 (5), 1305–1320.

[58] Shepard, D., 1968. A two-dimensional interpolation function for irregularly spaced data: Proceedings of the 23rd ACM national conference, New York, USA, 517–524.

[59] Shieds, D., Leonard, A., 2001. Investigation of a drag reduction on a circular cylinder in rotary oscillation. J. Fluid Mech. 431, 297–322.

[60] Timmermans, L. J. P., Minev, P. D., Vosse, F. N. V. D., 1996. An approximate projection scheme for incompressible flow using spectral elements. Int. J. Numer. Meth. Fluids 22, 673–688.

[61] Trefethen, L. N., Weideman, J. A. C., 2014. The exponentially convergent trapezoidal rule. SIAM Review 56, 385–458.

[62] Vacondio, R., Rogers, B., Stansby, P., Mignosa, P., Feldman, J., 2013. Variable resolution for SPH: A dynamic particle coalescing and splitting scheme. Comput. Methods Appl. Mech. Engrg. 256, 132–148.

[63] Van der Vorst, H. A., 1992. Bi-CGSTAB: A fast and smoothly converging variant of Bi-CG for the solution of nonsymmetric linear systems. SIAM J. Sci. and Stat. Comput. 13, 631–644.

[64] Violeau, D., Issa, R., 2007. Numerical modelling of complex turbulent free-surface flows with the SPH method: an overview. Int. J. Numer. Meth. Fluids 53, 277–304.

[65] Winckelmans, G., Leonard, A., 1993. Contributions to vortex particle methods for the computation of three-dimensional incompressible unsteady flows. J. Comput. Phys. 109 (2), 247 – 273.

[66] Xu, R., Stansby, P. K., Laurence., D., 2009. Accuracy and stability in incompressible SPH (ISPH) based on the projection method and a new approach. J. Comput. Phys. 228, 6703–6725.

[67] Zhang, J., Jackson, T. L., 2009. A high-order incompressible flow solver with WENO. J. Comput. Phys. 228 (7), 2426 – 2442.