We present a methodology for simulating three-dimensional flow of incompressible viscoplastic fluids modelled by generalised Newtonian rheological equations in non-trivial domains. It is implemented in a highly efficient framework for massively parallelisable computations on structured grids. In order to simulate flow in domains with non-trivial geometries, an embedded boundary approach is utilised, enabling a wide range of flow problems to be solved accurately and efficiently. This constitutes the first published implementation of embedded boundaries for simulating flow of viscoplastic fluids. The underlying algorithm employs a two-stage Runge-Kutta method for temporal discretisation, in which viscous terms are treated semi-implicitly and projection methods are utilised to enforce the incompressibility constraint. We augment the embedded boundary algorithm to deal with the variable apparent viscosity of the fluids. Since it depends strongly on the strain rate tensor, special care has been taken to approximate the components of the velocity gradients robustly near boundary cells, both for viscous wall fluxes in cut cells and for updates of apparent viscosity in cells adjacent to them. After validating the code against standard test cases, we demonstrate its capabilities by simulating fully three-dimensional creeping flow of Bingham plastics around objects moving through them for the first time. Our results shed new light on the flow fields around these objects.

Keywords: Embedded boundaries, non-Newtonian flow, high performance computing

---

a) Electronic mail: knutsverdrup@gmail.com
I. INTRODUCTION

In a recent publication, we presented high-performance software capable of simulating three-dimensional flow of viscoplastic fluids in time, utilising structured adaptive mesh refinement (AMR)\textsuperscript{1}. There, the efficacy of highly parallelisable structured meshing was demonstrated, though computational domains were restricted to rectangular and right rectangular prisms in two and three dimensions, respectively. In this work, we extend the software package to non-trivial geometries through the use of embedded boundaries (EBs). This was a natural extension of our work, and the embedded boundary approach was particularly well-suited to the structured mesh discretisation. Additionally, the AMR capabilities of the underlying software suite, AMReX, go hand in hand with embedded boundaries, allowing viscous boundary layers to be fully resolved through localised mesh refinement.

Although a lot of effort has been made towards simulations of viscoplastic fluids, their numerical treatment is by nature much more expensive than that of fluids which do not exhibit a yield stress\textsuperscript{2,3}. Consequently, researchers interested in these flows are often limited by the computational cost associated with simulating them, resulting in the majority of such work being restricted to two-dimensional and/or steady-state problems. This is especially true for fluid-structure interaction problems with many particles, such as the recent work by Koblitz et al.\textsuperscript{4}, which would not be practically possible within current software packages without considering the 2D steady-state problem. Having said that, there are notable advances being made, such as a study on the transition to turbulence of viscoplastic fluids past a cylinder in 3D, which was simulated using Papanastasiou regularisation for small Bingham numbers\textsuperscript{5}. A recent example of unsteady flow with fluid particle interactions is the study of time-dependent hydrodynamic interaction of 2D particles by Chaparian et al.\textsuperscript{6}. It was the goal of our previous publication to make progress towards more efficient and scalable simulations. We demonstrated results from large-scale simulations of time-dependent three-dimensional flows with yield stress playing a dominant role\textsuperscript{1}. For a full discussion of the development of numerical tools for viscoplastic flow problems, we refer to that paper and references therein.

There are many routes available for accommodating numerical simulations of flow in and around non-trivial geometries. The first factor to take into consideration is whether the underlying grid is structured or unstructured. Both types of grids are widely utilised. An
overview over the usage of each in published simulations of yield stress fluids is provided in the review by Saramito and Wachs\textsuperscript{2}. By definition, the difference between structured and unstructured grids is that the former requires regular connectivity of cells, while the latter does not. Consequently, the two types of grids have different advantages. Unstructured grids allow the use of curvilinear body-fitted meshes, which align with the geometry of the domain boundary. Structured grids, on the other hand, lead to highly efficient data storage and management. Embedded boundaries allow the incorporation of non-trivial geometries even though the underlying grid is structured. The methodology has several attractive qualities: rapid mesh generation regardless of complex geometries\textsuperscript{7}; avoidance of locally skewed grids; inherent compatibility with quad- and octree adaptive mesh refinement; and retainment of the efficient and user-friendly data storage associated with structured grids. Designed for computational speed and robustness, the AMReX framework facilitates efficient usage of Cartesian structured grids, and has extensive built-in support for EBs. We thus saw an opportunity to implement the first viscoplastic flow solver within such a framework, which one would expect to perform very well\textsuperscript{8}.

In the EB approach, non-Cartesian geometries are cut out of the underlying grid by storing local data representing the interface within each cell containing part of the geometry. There is thus a sharp interface represented by a plane in each rectangular-prism-shaped cell. Special consideration must then be taken to augment computational stencils and other algorithmic tools for these cut cells, in contrast to the uncovered or “normal” cells. This technique offers a rapid and relatively simple way of incorporating non-trivial geometries without sacrificing the efficiency associated with structured grids.

Over the last decades, the method has matured and become a robust tool. The idea was first used by Purvis and Burkhalter in 1979\textsuperscript{9}, and subsequently, Wedan and South\textsuperscript{10} to solve potential flow problems. Throughout the 1980s, the method was extended to solve the compressible Euler equations\textsuperscript{11–13}, which forms the basis for the general hyperbolic treatment of Colella et al. which we follow\textsuperscript{14}. Since we are studying an incompressible system, we also require the solution of Poisson equations, which was published by Johansen and Colella in 1998\textsuperscript{15}, a feat which they and McCorquodale naturally extended to solutions of the heat equation a few years later\textsuperscript{16}. Developing novel and improved schemes for embedded boundaries is still an active area of research\textsuperscript{17–19}.

This paper constitutes an entirely novel utilisation of EB techniques to treat flow prob-
lems involving generalised Newtonian fluids, and more specifically yield-stress fluids. While evaluating the software, the extension has allowed us to obtain rich insight into the flow fields around objects moving through Bingham fluids. In particular, the three-dimensional effects on the yield surface of a sphere in such a configuration are properly investigated for the first time, and we show how more general, asymmetric flows can be simulated just as easily.

In section II we will formulate the mathematical description of the system of governing partial differential equations and discuss relevant fluid rheologies. Section III is devoted to the numerical algorithm employed to simulate the fluid flow, including treatment of embedded boundaries. Thorough validation is performed in section IV before we evaluate the code for more demanding problems with genuinely three-dimensional effects in section IV B. Section V concludes the article.

II. MATHEMATICAL FORMULATION

Our domain $\Omega \subset \mathbb{R}^3$ has a boundary denoted by $\Gamma = \partial \Omega$, with inward pointing unit normal vector $\hat{n}$. We take the gradient of a vector $u$ as the tensor with components

$$(\nabla u)_{ij} = \frac{\partial u_j}{\partial x_i}, \quad (1)$$

while the divergence of a rank-2 tensor field $\tau$ is defined such that

$$(\nabla \cdot \tau)_j = \sum_{i=1}^{3} \frac{\partial \tau_{ij}}{\partial x_i}. \quad (2)$$

Variables are functions of position $x \in \Omega$ and time $t \geq 0$. We denote by $\rho \in \mathbb{R}$ the material density. The velocity field is introduced as $u(x, t) \in \mathbb{R}^3$, with components $u$, $v$ and $w$. The Cauchy stress tensor $\sigma(x, t)$ is defined as the sum of isotropic and deviatoric parts, $\sigma = -p \mathbf{I} + \tau$. Here, the pressure $p(x, t) \in \mathbb{R}$ is multiplied by the identity tensor, while the deviatoric part of the stress tensor is denoted $\tau(x, t) \in \mathbb{R}^{3 \times 3}_{\text{sym}}$. 
A. Governing partial differential equations

We consider time-dependent flow of incompressible, generalised Newtonian fluids. Denoting external body forces by \( f \), the relevant governing equations are

\[
\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = \frac{1}{\rho} \left( -\nabla p + \nabla \cdot \boldsymbol{\tau} + f \right) \quad \text{in } \Omega \quad (3a)
\]

\[
\nabla \cdot \mathbf{u} = 0 \quad \text{in } \Omega \quad (3b)
\]

\[
\boldsymbol{\tau} = \eta (\|\dot{\gamma}\|) \dot{\gamma} \quad \text{in } \Omega \quad (3c)
\]

\[
\mathbf{u} = \mathbf{u}_{BC} \quad \text{on } \Gamma \quad (3d)
\]

where the strain rate tensor \( \dot{\gamma} = \nabla \mathbf{u} + (\nabla \mathbf{u})^\top \) is (twice) the symmetric part of the velocity gradient, and \( \|\cdot\| \) denotes the scaled Frobenius norm such that \( \|\dot{\gamma}\| = \sqrt{\frac{1}{2} \text{tr} (\dot{\gamma} \dot{\gamma}^\top)} \), as is customary for convenience in viscoplastic fluid mechanics. Equation \((3a)\) is Cauchy’s momentum balance, \((3b)\) is the incompressibility constraint, and \((3c)\) relates the stress to the specific rheological equation for a generalised Newtonian fluid model through the apparent viscosity \( \eta (\|\dot{\gamma}\|) \).

B. Rheology

An applied shear strain acting on a fluid causes flow through viscous deformation, and rheological equations give the relationship between the stress and strain tensors, their temporal derivatives and physical variables such as time, temperature and pressure\(^2\). As seen from \((3c)\), we restrict ourselves to relatively simple equations of state on the form \( \boldsymbol{\tau} = \boldsymbol{\tau} (\dot{\gamma}) \). In other words, the stress response is solely dependent on the strain rate tensor. Furthermore, the relation is characterised by an apparent viscosity \( \eta \), which is a function of the second invariant of the strain rate tensor, \( \text{tr} (\dot{\gamma} \dot{\gamma}^\top) \). For Newtonian fluids, a constant viscosity coefficient quantifies the proportionality between stress and strain rate. Due to the analogy to this simplest of viscous models, the fluids which we consider are called generalised Newtonian fluids.

Although generalised Newtonian fluids cannot capture exotic rheological phenomena such as thixotropy and rheopecty\(^2\)^{21\,22}, they can still capture a rich variety of fluid dynamics. Physical phenomena which are widely encountered in applications include shear induced thinning and thickening, and the existence of a yield stress, below which flow does not occur.
In table I, we list the fluids models which have been implemented in our software package, and relevant references. Newtonian fluids have a constant dynamic viscosity coefficient $\mu$, while shear thinning and thickening is modelled through the power law model of Ostwald and de Waele, in which $\mu$ is replaced by the consistency $\kappa$ and a flow index $n$ is introduced. Its value is between zero and one for shear thinning fluids (pseudoplastics) and larger than one for shear thickening fluids (dilatants). For $n = 1$, we recover the Newtonian model.

Viscoplastics, commonly referred to as yield stress fluids, are characterised by a yield stress $\tau_0$, which is a threshold value that must be overcome by the applied shear stress for any flow to occur at all. Since viscoplasticity is the most interesting feature in our fluid models, and at the same time the most challenging to capture numerically, we will be using the simplest such model to validate and showcase our software. It is the Bingham plastic fluid, which has a linear relationship between stress and strain-rate above the yield limit:

$$
\begin{align*}
\|\tau\| & \leq \tau_0, & \text{if } \dot{\gamma} = 0, \\
\tau &= (\mu + \frac{\tau_0}{\|\dot{\gamma}\|}) \dot{\gamma}, & \text{if } \dot{\gamma} > 0.
\end{align*}
$$

(4)

The Bingham plastic rheological model thus separates the flow into two separate states. We refer to the case of zero strain-rate as an unyielded state, while viscous flow occurs for the yielded state, when $\|\tau\| > \tau_0$. For the yielded Bingham fluid, we can rewrite (4) as a generalised Newtonian fluid, taking the apparent viscosity function as

$$
\eta = \mu + \frac{\tau_0}{\|\dot{\gamma}\|}.
$$

(5)

Since this results in a singularity in the limit of zero strain rate, we utilise the regularisation approach introduced by Papanastasiou. By introducing a small regularisation parameter $\varepsilon$ and multiplying the singular term by one minus a decaying exponential, we remove the singularity. Consequently, we limit the maximum possible viscosity in the fluid and obtain an apparent viscosity function defined for all values of the rate-of-strain tensor,

$$
\eta = \mu + \frac{\tau_0}{\|\dot{\gamma}\|} \left(1 - e^{\|\dot{\gamma}\|/\varepsilon}\right).
$$

(6)

The Herschel-Bulkley model generalises power-law fluids to the realm of viscoplasticity in the same way as the Bingham plastic generalises Newtonians. Finally, we include a model by de Souza Mendes and Dutra, which has the same flexibility and desirable attributes as the regularised Herschel-Bulkley model, but which seeks to account for the limiting behaviour...
Fluid model & $\eta(\|\dot{\gamma}\|)$
\hline
Newtonian & $\mu$ \\
Power-law & $\kappa \|\dot{\gamma}\|^{n-1}$ \\
Bingham & $\mu + \frac{\tau_0}{\|\dot{\gamma}\|} (1 - e^{-\|\dot{\gamma}\|/\epsilon})$ \\
Herschel-Bulkley & $\kappa \|\dot{\gamma}\|^{n-1} + \frac{\tau_0}{\|\dot{\gamma}\|} (1 - e^{-\|\dot{\gamma}\|/\epsilon})$ \\
Souza Mendes-Dutra & $\kappa \|\dot{\gamma}\|^{n-1} + \frac{\eta_0}{\|\dot{\gamma}\|} (1 - e^{-\eta_0\|\dot{\gamma}\|/\tau_0})$ \\
\hline

**TABLE I: Generalised Newtonian fluid models available in the fluid solver.**

...at small strain rate as a physical phenomena, rather than as a convenient numerical tool. For a further discussion on viscoplastic fluids and their numerical treatment, we refer to our previous paper [1].

### III. NUMERICAL ALGORITHM

This section is devoted to the details of the numerical algorithm utilised in order to solve (3) efficiently, taking advantage of modern supercomputer architectures. The basis for the fluid solver is a projection method for solving the variable density incompressible Navier-Stokes equations on an adaptive mesh hierarchy, as described for the constant viscosity case by Almgren et al. [29], with extensions to low Mach number reacting flows with temperature-dependent viscosity provided by Pember et al. [30] and Day and Bell [31], among others. By using the software framework [AMReX], the resulting implementation is such that the code can be run on architectures from single-core laptops through to large-scale distributed cluster computers.

[AMReX] is a mature, open-source software framework for building massively parallel structured AMR applications. AMReX contains extensive software support for explicit and implicit grid-based operations. Multigrid solvers, including those for tensor systems, are included for cell-based and node-based data. AMReX uses a hybrid MPI/OpenMP approach for parallelisation; in this model individual grids are distributed to MPI ranks, and OpenMP is used to thread over logical tiles within the grids. Applications based on AMReX have demonstrated excellent strong and weak scaling up to hundreds of thousands of cores [32-35]. For a demonstration of scalability in the specific context of yield stress fluids, we refer to...
our previous paper on the subject\cite{1}.

The functionality provided by the AMReX framework facilitates simple solutions to many of the complex subproblems we encounter. For example, all linear systems solved in our software utilises AMReX' built-in multilevel geometric multigrid solver\cite{35} for EB systems, with the biconjugate gradient stabilised method (BiCGSTAB)\cite{37} as the base algorithm for the linear solver.

For the sake of brevity, we omit a full, low-level description of the algorithmic implementation and point the interested reader to the code repository\cite{38}, and the documentation for the underlying software framework AMReX\cite{39}.

A. Incompressible flow solver

For the sake of clarity, we omit treatment of boundaries and communication across refinement levels in the following description of the incompressible flow solver. Instead, we focus on the steps required to move the system forward one time step on a single refinement level. Although the software is general enough to handle variable-density incompressible flows through a conservative advection of $\rho$, we only consider cases where it is constant, and consequently do not discuss conservative density updates, which in any event are trivial compared to the velocity terms. Note also that there are no external forces for the cases in this paper, so $\mathbf{f} = 0$ throughout.

We follow a method-of-lines (MOL) approach, in which the momentum equation as given by (3a) is only discretised in time, so that Runge-Kutta type schemes for ordinary differential equations (ODEs) can be utilised for the temporal advancement. We affix a superscript to the simulated variables at the $m$-th time step, so that the time itself is $t^m$ and the velocity profile $\mathbf{u}^m$. Algorithm 1 gives the pseudocode to advance the simulation from time step $m$ to $m + 1$. 

8
Algorithm 1: Advance simulation one time step

1: function ADVANCE\((t^m, \mathbf{u}^m, p^m)\)
2: \hspace{1em} APPLYBOUNDARYCONDITIONS(\Gamma)
3: \hspace{1em} \Delta t := \text{TIMESTEPSIZE}(t^m, \mathbf{u}^m, p^m)
4: \hspace{1em} procedure PREDICTOR
5: \hspace{2em} C^m, \eta^m, V^m := \text{DERIVEDQUANTITIES}(\mathbf{u}^m)
6: \hspace{2em} C \leftarrow C^m
7: \hspace{2em} \eta \leftarrow \eta^m
8: \hspace{2em} V \leftarrow V^m
9: \hspace{2em} \psi \leftarrow \rho \mathbf{u}^m + \Delta t (\rho C + V - \nabla p^m + f)
10: \hspace{2em} \mathbf{\bar{u}}^* := \text{UPDATEVELOCITY}(\Delta t, \eta, \psi)
11: \hspace{2em} \mathbf{u}^*, p^* := \text{NODALPROJECTION}(\Delta t, \mathbf{\bar{u}}^*)
12: \hspace{1em} end procedure
13: \hspace{1em} procedure CORRECTOR
14: \hspace{2em} C^*, \eta^*, V^* := \text{DERIVEDQUANTITIES}(\mathbf{u}^*)
15: \hspace{2em} C \leftarrow \frac{1}{2} (C^m + C^*)
16: \hspace{2em} \eta \leftarrow \frac{1}{2} (\eta^m + \eta^*)
17: \hspace{2em} V \leftarrow \frac{1}{2} (V^m + V^*)
18: \hspace{2em} \psi \leftarrow \rho \mathbf{u}^m + \Delta t (\rho C + V - \nabla p^* + f)
19: \hspace{2em} \mathbf{\bar{u}}^{m+1} := \text{UPDATEVELOCITY}(\Delta t, \eta, \psi)
20: \hspace{2em} \mathbf{u}^{m+1}, p^{n+1} := \text{NODALPROJECTION}(\Delta t, \mathbf{\bar{u}}^{m+1})
21: \hspace{1em} end procedure
22: \hspace{1em} t^{m+1} := t^m + \Delta t
23: \hspace{1em} m \leftarrow m + 1
24: end function

After applying boundary conditions and computing the time step size according to the formula derived by Kang, Fedkiw and Liu (or setting it to a fixed value), we perform a two-step Runge-Kutta scheme with a predictor and a corrector step. The pseudocode in each of the procedures PREDICTOR and CORRECTOR has been written so as to highlight the similarities between them. Although we are dealing with a multi-step algorithm including
projections within each procedure, this elucidates the classic Runge-Kutta scheme underlying
the temporal integration.

Before going into the specifics of the algorithm substeps, we need to mention the spatial
discretisation of our physical variables. Velocities and derived quantities are all computed
at cell centres, while the pressure is computed at nodes. Consequently, pressure gradients
are stored at cell centres, and can be added directly in the momentum balance.

The first step within each of the procedures is to compute the quantities which are
directly derived from the velocity field and its gradients. The relevant pseudocode is given
in algorithm 2. In order to accurately capture convection within the fluid, a slope-based
upwinding procedure is utilised. Velocity slopes are computed within each cell by comparing
values in neighbouring cells. These slopes are then used to extrapolate the input velocity
to cell faces, which allows us to compute the convective term $\mathbf{C} = -\mathbf{u} \cdot \nabla \mathbf{u}$. Before doing
this, however, we enforce the incompressibility constraint in (3b) by projecting the face-
based extrapolated velocity $\tilde{\mathbf{u}}$ onto the space of solenoidal vector fields. This is necessary
since the values extrapolated by upwinding the slopes are not necessarily divergence-free.

The Helmholtz decomposition allows us to write any vector field as a sum of solenoidal and
irrotational parts, so we define

$$\tilde{\mathbf{u}} = \mathbf{u} + \nabla \phi,$$  \hspace{1cm} (7)

where $\mathbf{u}$ has zero divergence and $\nabla \phi$, being a scalar gradient, is irrotational. Taking the
divergence of (7) leads to a Poisson equation for $\phi$,

$$\nabla^2 \phi = \nabla \cdot \tilde{\mathbf{u}},$$  \hspace{1cm} (8)

which is straightforward to solve and allows us to compute the divergence-free velocity field
$\mathbf{u}$. Since the three components of $\mathbf{u}$ are extrapolated to separate faces, the unknown
scalar $\phi$ must be cell-centred in order for its gradients to end up on the corresponding faces.
For this reason, we refer to this projection as the cell-centred projection, although it has
previously been referred to as the MAC projection due to its historical links to the spatial
discretisation in the original marker and cell method.\[10\]
Algorithm 2 Compute derived quantities: convective term, apparent viscosity and explicit viscous term

1: function DerivedQuantities(u)
2:     s = ComputeSlopes(u)
3:     \( \hat{u}^F = \text{ExtrapolateToFaces}(u, s) \)
4:     u^F = \text{CellCentredProjection}(\hat{u}^F)
5:     C = -u \cdot \nabla u^F \quad \triangleright \text{Convective term}
6:     \|\dot{\gamma}\| = \|\nabla u + (\nabla u)^T\| \quad \triangleright \text{Strain rate magnitude}
7:     \eta = \eta(\|\dot{\gamma}\|) \quad \triangleright \text{Apparent viscosity}
8:     V = \frac{1}{\rho} \nabla \cdot \left( \eta (\nabla u)^T \right) \quad \triangleright \text{Non-linear stress divergence}
9:     return C, \eta, V
10: end function

With the convective terms in place, the next step is to compute the viscous terms. Since viscous terms dominate in many of the problems we are interested in, and since they can lead to overly restrictive time step criteria when treated explicitly, we opt for a semi-implicit temporal discretisation. Noting that the stress divergence is

\[
\nabla \cdot \tau = \nabla \cdot (\eta \dot{\gamma}) = \nabla \cdot (\eta \nabla u) + \nabla \cdot \left( \eta (\nabla u)^T \right),
\]

and thus contains one term with purely unmixed derivatives in \( u \), and one with mixed and transverse terms, we treat the former implicitly and the latter explicitly. This is due to the fact that the unmixed result is of a compatible layout with the temporal derivative of \( u \), and can readily be set up for a linear solve. The explicit viscous term, derived from the input velocity is therefore

\[
V = \frac{1}{\rho} \nabla \cdot \left( \eta (\nabla u)^T \right),
\]

where \( \eta \) (via \( \|\dot{\gamma}\| \)) is also calculated from the input velocity, as shown on lines 6-8 in algorithm 2. Note that one of the rheological equations in table I must be specified for the simulation.

In the predictor, the derived quantities from the input velocity \( u^m \) are used directly in the velocity update. In the corrector, on the other hand, the derived quantities are first computed from the predictor’s output velocity \( u^* \), before the average is calculated and utilised in the update. See lines 6-8 and 15-17 in algorithm 1. The semi-implicit velocity
update consists of solving the system
\[
(\rho - \Delta t \nabla \cdot (\eta \nabla)) \tilde{u} = \rho u + \Delta t \left( \rho C + V - \nabla p + f \right)
\]  
(11)

with respect to \( \tilde{u} \), the new-time velocity.

It is necessary to apply another projection in order to enforce the incompressibility constraint for the new velocity. Conveniently, we can update the pressure at the same time. The two equations
\[
\begin{align*}
\frac{\tilde{u} - u^{\text{old}}}{\Delta t} &= -u \cdot \nabla u + \frac{1}{\rho} (\nabla \cdot \tau + f) \\
\frac{u^{\text{new}} - \tilde{u}}{\Delta t} &= -\frac{1}{\rho} \nabla p
\end{align*}
\]  
(12a)
(12b)

sum to a MOL discretised version of (3a), and the latter is of the form of a Helmholtz decomposition, just like (7). We therefore add the pressure gradient term back to the new-time velocity,
\[
\tilde{u} \leftarrow \tilde{u} + \Delta t \frac{\nabla p}{\rho}
\]  
(13)

before solving the Poisson equation
\[
\nabla \cdot \left( \frac{\Delta t}{\rho} \nabla p^{\text{new}} \right) = \nabla \cdot \tilde{u},
\]  
(14)

for \( p^{\text{new}} \), and obtain the new, divergence-free velocity field
\[
u^{\text{new}} = \tilde{u} - \Delta t \frac{\nabla p^{\text{new}}}{\rho}.
\]  
(15)

Note that in this projection, all velocity components are stored on cell centres, and the pressure is thus nodal. Consequently, we refer to it as the nodal projection. It is in fact a second-order accurate approximate projection method, the likes of which have been thoroughly analysed by Almgren et al.\textsuperscript{42} The predictor outputs the velocity field \( u^{*} \) and pressure \( p^{*} \), while for the corrector the corresponding variables are \( u^{m+1} \) and \( p^{m+1} \).

After the corrector, \( \Delta t \) is added to the current time, and \( m \) is incremented, before continuing on to the next time step.

\section*{B. Embedded boundaries}

The EB approach allows us to retain the structured adaptive mesh which AMReX is built for, while simulating flow in non-trivial domain boundaries. An arbitrary implicit
signed distance function is used to describe the geometry, which is then discretised as planar intersections with each cell. The intersections are continuous at cell faces, which means that they are piecewise linear everywhere. Cells are identified by their index vector \( \mathbf{i} = (i, j, k) \).

We store a flag in each cell marking it as either uncovered (normal), covered (ignored), or cut. In the latter case, additional data is stored within the cell, so that we can take geometrical information into account for computations in that cell. Since we only consider single-valued cut cells, only four numbers are necessary to uniquely define the cut cell, namely the three components of the boundary surface unit normal \( \hat{\mathbf{n}}_{EB} \) plus the volume fraction \( \alpha \in (0,1) \) of fluid within the cell. We orient \( \hat{\mathbf{n}}_{EB} \) so that it points into the fluid domain, and let \( \alpha = 0 \) for covered cells and \( \alpha = 1 \) for uncovered ones. Although these quantities describe the cut cell, several helpful additions are available in the EB framework provided by AMReX, such as the uncovered area fractions on each face (including the EB), the volume centroid of the fluid, and the area and centroid of the embedded boundary. Figure 1 illustrates the cut cell and the relevant quantities for our computations. For details on the EB implementation, we refer to the AMReX documentation, and for an explanation of how EB data is computed from a signed distance function, the reader is referred to Gokhale et al.\textsuperscript{19}

All variables in our simulations are stored at cell centres, even if these are covered by the embedded boundary. This has the strong advantage of allowing us to avoid altering computational stencils for calculating extrapolated values on faces or approximating gradients based on neighbouring cells which are cut. We only need to worry about the cut cells themselves, which can have covered (and thus undefined) neighbours. The EB tools built in to the linear solvers for systems of linear equations (solutions to the Poisson equations) are based on the work of Johansen and Colella.\textsuperscript{15}

1. **Flux computations**

Our main challenge algorithmically (apart from the linear solves, for which AMReX has built-in EB support) is to successfully compute the derived quantities within each cut cell, i.e. the convective term \( C \), the rate-of-strain tensor magnitude \( \|\dot{\gamma}\| \), the apparent viscosity \( \eta \) and the viscous term \( V \). It is crucial that this is done in a manner which avoids time step constraints resulting from small cut cell volumes.
FIG. 1: Schematic illustration of the Cartesian cut cell viewed in 3D (left) and through a slice with constant $z$ (right). The plane representing the embedded boundary is marked in red, and its surface normal vector $\mathbf{\hat{n}}$ points from the covered solid region to the uncovered fluid domain with volume fraction $\alpha$. The right plot also shows the relevant surface areas.

Let us first consider terms which can be written as the divergence of a flux $\mathbf{F}$, i.e.

$$D = \nabla \cdot \mathbf{F}.$$  \hspace{1cm} (16)

For these, we utilise the flux redistribution technique for embedded boundaries as developed by Colella et al.\cite{Colella1989, Colella1991}. Applying the divergence theorem to a cut cell control volume, we find that

$$D_i^C = \frac{1}{V_i} \int_{\Omega_i} \nabla \cdot \mathbf{F} \, dV = \frac{1}{V_i} \int_{\partial \Omega_i} \mathbf{F} \cdot d\mathbf{A} = \frac{1}{V_i} \sum_{f=1}^{N_i} A_f F \cdot \mathbf{\hat{n}}_f$$ \hspace{1cm} (17)

is a conservative estimate of $D$. Here, $V_i = \alpha_i \Delta x \Delta y \Delta z$ is the cell volume, $N_i$ is the number of faces (EB or otherwise) enclosing the cell, and $A_f$ is the uncovered surface area of the face $f$. For all non-EB faces, we can evaluate the flux tensor $\mathbf{F}$ just as for uncovered cells, since physical variables are stored at cell centres. Consequently, the slope computation and upwinding procedures do not need to be altered for EB cells except that one-sided upwinding is applied in the case of covered neighbouring cells. In other words, we just need to use the
FIG. 2: Decomposition of fluxes employed in the cell-averaged evaluation of divergence. Summation of these fluxes multiplied by cell surface areas leads to a conservative estimate of the divergence of the flux field.

EB information to extrapolate $F$ to the uncovered face centroid and multiply it by the corresponding face area. Subscripting fluxes and area fractions in direction $d$ by $+$ at one end and $-$ at the other, Eq. (17) can be written

$$ D^C_i = \frac{1}{V_i} \sum_{d \in \{x,y,z\}} (F_{d,+}A_{d,+} - F_{d,-}A_{d,-}) \cdot \hat{n}_d + \frac{1}{V_i} F_{EB} A_{EB} \cdot \hat{n}_{EB}, $$  

and the only special consideration required is the evaluation of the flux tensor at the EB centroid, $F_{EB}$. Figure 2 displays these fluxes in a 2D slice of the cut cell.

The downside of the conservative flux as computed in Eq. (18) is that the so-called small cell problem arises in the explicit temporal discretisation. The time step size is restricted since it is proportional to $\alpha$, which can be arbitrary small in cut cells. In order to circumvent this, we define a non-conservative approximation to the divergence as an average of the conservative approximations in the neighbourhood of the cell,

$$ D^{NC}_i = \frac{\sum \alpha_{i'} D^C_{i'}}{\sum \alpha_{i'}}. $$  

We define the neighbourhood as the set

$$ \mathcal{N}(i) = \{ i' \in \mathbb{Z}^3 : \min|i - i'| = \max|i - i'| = 1, \alpha_{i'} > 0 \}, $$

i.e. all uncovered or cut cells whose index vector components differ by at most one from $i$, except cell $i$ itself. A linear hybridisation of the conservative and non-conservative flux
approximations is then given by

\[ D_i^H = \alpha_i D_i^C + (1 - \alpha_i) D_i^{NC}. \] (21)

Note that this hybrid flux approximation has the desired behaviour in the limits of \( \alpha_i \in [0, 1] \), and removes the effect of the local volume fraction on the time step size.

Although the hybrid flux stabilises the CFL restriction, it does not strictly enforce conservation. The excess material lost or gained due to the usage of (21) rather than (18) is

\[ \delta D_i = \alpha_i (D_i^C - D_i^H) = \alpha_i (1 - \alpha_i) (D_i^C - D_i^{NC}), \] (22)

and this excess must be redistributed back to the cell neighbours. This is done with the use of weights \( w_{i,i'} \geq 0 \) which quantify the fraction of \( \delta D_i \) redistributed to cell \( i' \), and which must satisfy \( \sum_{i' \in N(i)} w_{i,i'} \alpha_{i'} = 1 \) for strict conversation. We use the simple weights

\[ w_{i,i'} = \frac{1}{\sum_{i' \in N(i)} \alpha_{i'}}, \] (23)

which are actually independent of \( i' \). The final, discrete approximation to the divergence operator is thus

\[ D_i = D_i^H + \sum_{i' \in N(i)} w_{i',i} \delta D_{i'}. \] (24)

2. Convective term

By the chain rule, we have

\[ \nabla \cdot (u \otimes u^F) = u \cdot \nabla u^F + (\nabla \cdot u) u^F = u \cdot \nabla u^F, \] (25)

where the last equality holds due to the incompressibility constraint in (3b). Seeing the convective term as the divergence of a tensor flux, \( C = \nabla \cdot F^C \), we can apply the procedure outlined above in order to compute the convective term in EB cells. Note that we enforce an inhomogeneous Dirichlet condition on \( u \) at all EB walls, so that the components of the convective flux tensor are all zero there: \( F_{EB}^C = u \otimes u^{EB} = 0 \). As such, we do not need to make any special considerations for the convective fluxes at embedded boundaries. This is in contrast to the viscous wall fluxes, which we will deal with next.
FIG. 3: Computation of viscous wall fluxes at the EB centroid of cell \( i, j, k \). The red line represents the embedded boundary, which cuts the cell such that its surface centroid and normal are \( b \) and \( \hat{n} \), respectively. In this example, the largest component of the surface normal is in the \( x \)-direction, so we interpolate to the points marked with crosses in the \( yz \)-planes corresponding to \( i + 1 \) and \( i + 2 \).

3. Viscous term

The flux tensor arising from the viscous term is \( F^V = \eta (\nabla u)^T \), which can be non-zero on the EB surfaces. We therefore need a procedure to compute the viscous wall flux at the EB surface centroid \( b \), which is given relative to local cut cell coordinates, where the cell centre is the origin. To this end, we compute the gradient of each velocity component along the EB surface normal vector \( \hat{n}_{EB} \). In order to achieve this, we utilise AMReX’ built-in biquadratic interpolation routine to find the value of \( u \) at two points located at distances \( d_1 \) and \( d_2 \) from the EB surface centroid \( b \). Figure 3 illustrates the interpolation points for an example cut cell.

We start by finding which is the largest component of the surface normal vector. The biquadratic interpolation will be done in planes where the corresponding coordinate is held fixed. Consider the case when \( \max(n_x, n_y, n_z) = n_x \), as in figure 3. In order to make sure that we are moving away from the EB, we let \( s = \text{sign}(n_x) \) and define the interpolation points
as those where the line \( b + d \hat{n} \) intersect the planes \( x = s \) and \( x = 2s \). The corresponding distances from \( b \) are
\[
d_1 = \frac{s - b_x}{n_x}, \quad d_2 = \frac{2s - b_x}{n_x}. \tag{26}
\]

We can thus find the \( y \) and \( z \) coordinates, and utilise biquadratic interpolation to obtain velocity values based on the 9 nearest points in the plane. Denoting the interpolated velocities by \( u_1 \) and \( u_2 \), and allowing a prescribed velocity \( u_{ EB } \) on the boundary (zero throughout this paper), (A2) gives the normal derivative as
\[
\frac{\partial u}{\partial \hat{n}} \bigg|_{ EB } = \frac{d_2^2(u_1 - u_{ EB}) - d_1^2(u_2 - u_{ EB})}{d_1 d_2(d_2 - d_1)} = \frac{d_2^2 u_1 - d_1^2 u_2}{d_1 d_2(d_2 - d_1)}. \tag{27}
\]

All components of the velocity gradient are now available by taking the projections of \( \frac{\partial u}{\partial \hat{n}} \) in the relevant Cartesian direction, i.e. multiplying by the corresponding component of \( \hat{n} \). We can thus compute \( \dot{\gamma}, \eta \) and finally
\[
\mathbf{F}^V = \eta (\nabla \mathbf{u})^\top \tag{28}
\]
in the cut cell.

4. Strain-rate tensor and apparent viscosity

Since our aim is to accurately capture the flow patterns of fluids with apparent viscosity functions which depend strongly on the magnitude of the rate-of-strain tensor, it is essential that we can compute the components of the velocity gradient tensor to second-order accuracy. The procedure outlined for the viscous wall fluxes above is tailored for computing the values on EB walls, but in the present case we require them at cell centres. In order to circumvent the problem of covered neighbour cells, we adjust the stencil used for difference estimation. Similarly to in the previous subsection, we consider a function \( y(x) \) whose values we know at three points \( x_i, i \in \{0, 1, 2\} \), with the alteration that \( x_0 \neq 0 \) but the points have equal spacing \( \Delta x \). According to [A2], the gradient of a quadratic polynomial fit to these points is given by
\[
\frac{\partial f}{\partial x} = \frac{1}{\Delta x^2} \left( \frac{1}{2}(x - x_1 + x - x_2)y_0 
- (x - x_2 + x - x_0)y_1 
+ \frac{1}{2}(x - x_0 + x - x_1)y_2 \right). \tag{29}
\]
We can thus evaluate the gradient of the function at the given points to second order accuracy with simple coefficients, since

\[
\frac{\partial f}{\partial x} \bigg|_{x_0} = \frac{1}{\Delta x} \left( -\frac{3}{2} y_0 + 2 y_1 - \frac{1}{2} y_2 \right) \quad (30a)
\]

\[
\frac{\partial f}{\partial x} \bigg|_{x_1} = \frac{1}{2\Delta x} (y_2 - y_0) , \quad (30b)
\]

\[
\frac{\partial f}{\partial x} \bigg|_{x_2} = \frac{1}{\Delta x} \left( \frac{1}{2} y_0 - 2 y_1 + \frac{3}{2} y_2 \right) . \quad (30c)
\]

In order to evaluate the velocity gradient tensor in cut cells, we simply check whether the neighbouring cells in each direction are covered, and if so, utilise a one-sided quadratic difference estimator rather than the central one. Consequently, we are only fishing for values in well-defined cells. With the estimates for velocity gradient components in place, we can evaluate the cell-centred rate-of-strain magnitude and apparent viscosity directly.

IV. VALIDATION

Before evaluating the methodology for genuinely three-dimensional viscoplastic flows, we need to ensure that the underlying incompressible flow solver has the desired order of accuracy by performing a grid convergence study for a problem with known solution. Furthermore, we verify that viscoplastic effects are captured by computing the solution to a problem which has an analytical solution for the Bingham model.

A. Convergence study: Taylor-Green vortices

In order to demonstrate spatio-temporal second order convergence of the presented algorithm, we consider the two-dimensional unsteady Taylor-Green vortices in a Newtonian fluid, for which an analytical solution exists. Note that when considering viscoplastic fluids, the accuracy depends on the Papanastasiou regularisation parameter, so a Newtonian benchmark is necessary to validate the accuracy of the spatio-temporal solver. This problem is a well-known test case in computational fluid mechanics, and consists of exponentially decaying vortices due to viscous dissipation. The computational domain is \( \Omega = [0, L]^2 \times [0, W] \), where \( W \) is the width to which we extend the test case in the out of plane dimension. Periodic boundary conditions are applied on all faces, and we take \( U \) as the maximum of
the initial velocity distribution. We obtain dimensionless variables by scaling lengths by \( L \), velocities components by \( U \), time by \( L/U \) and pressure by the kinetic energy density \( \rho U^2 \).

The Taylor-Green solution is then

\[
\hat{u} = \sin (2\pi \hat{x}) \cos (2\pi \hat{y}) e^{-8\pi^2 \hat{t}/Re}, \quad (31a)
\]
\[
\hat{v} = -\cos (2\pi \hat{x}) \sin (2\pi \hat{y}) e^{-8\pi^2 \hat{t}/Re}, \quad (31b)
\]
\[
\hat{p} = \frac{1}{4} (\cos (4\pi \hat{x}) + \cos (4\pi \hat{y})) e^{-16\pi^2 \hat{t}/Re}, \quad (31c)
\]

where we have introduced the Reynolds number

\[
Re = \frac{\rho UL}{\mu}. \quad (32)
\]

For a series of spatial resolutions, characterised by the amount of cells \( N \) in the \( x \)- and \( y \)-directions, our system is initialised to the distribution given by inserting \( \hat{t} = 0 \) into (31) with \( Re = 100 \), and advanced to the time \( \hat{t} = 1 \). The resulting velocity field is subtracted from the analytical solution in each cell, and we denote this residual \( \varepsilon_N(x) \). For two meshes with resolution \( N \) and \( 2N \), the convergence rate of our numerical method can be computed as

\[
\begin{align*}
\displaystyle r_* &= \log_2 \left\| \frac{\varepsilon_{2N}}{\varepsilon_N} \right\|_*, \\
\end{align*}
\]

where \( \| \cdot \|_* \) is an appropriate function norm, typically one of

\[
\begin{align*}
\| \varepsilon \|_1 &= \int_{\Omega} |\varepsilon(x)| \, dx, \quad (34a) \\
\| \varepsilon \|_2 &= \left( \int_{\Omega} |\varepsilon(x)|^2 \, dx \right)^{\frac{1}{2}}, \quad (34b) \\
\| \varepsilon \|_\infty &= \max_{\Omega} |\varepsilon(x)|. \quad (34c)
\end{align*}
\]

Table II shows how the convergence rate for our solver approaches two in all these norms as \( N \) grows, as expected.

**B. Poiseuille flow in a cylinder**

In order to validate both the viscoplastic rheology and the embedded boundaries, we consider Poiseuille flow of a Bingham fluid on the interior of a circular cylinder with radius \( R \). Poiseuille flow is driven by a constant pressure gradient \( \mathcal{G} \) applied in the direction
of flow, which in our case is along the $z$-axis. Since the problem is both axisymmetric and independent of $z$, the variation in velocity is a function of $r = \sqrt{(x-x_c)^2 + (y-y_c)^2}$ alone, where the cylinder centre axis has coordinates $(x_c, y_c, z)$. Scaling distances by $R$ and velocity by the maximum velocity $U$ in the centre of the cylinder, the analytical solution at steady-state is given by

$$\hat{w}(\hat{r}) = \begin{cases} 
1, & \hat{r} \leq \hat{r}_0 \\
1 - \left(\frac{\hat{r}-\hat{r}_0}{1-\hat{r}_0}\right)^2, & \hat{r} > \hat{r}_0
\end{cases}$$

(35)

Here, we have introduced the dimensionless radius

$$\hat{r}_0 = \frac{2\tau_0}{GR},$$

(36)

which separates the flow into yielded and unyielded regions, i.e. represents the yield surface. Note that this is the only free variable in the dimensionless system, and represents the relative strength of the yield stress to the applied pressure gradient. For the sake of completeness, we note that the characteristic velocity is given by

$$U = \frac{GR^2}{4\mu} (1 - \hat{r}_0)^2.$$  

(37)

Our simulations were performed in a domain of size $\Omega = [0, 2.5R]^2 \times [0, R]$ with the cylinder centred at $\hat{x}_c = \hat{y}_c = 1.25$. We set physical parameters of the applied pressure gradient and fluid such that $r_0 = \frac{1}{2}$.

Figure 4a shows the resulting velocity distribution in the $xy$-plane for a simulation with $N = R/\Delta x = 32$ cells over the radial distance, and regularisation parameter $\varepsilon = 0.01 \text{s}^{-1}$. In the middle of the cylinder, we can see the plug of unyielded fluid travelling with constant

| $N$ | $\|\varepsilon\|_1$ | $r_1$ | $\|\varepsilon\|_2$ | $r_2$ | $\|\varepsilon\|_\infty$ | $r_\infty$ |
|-----|-----------------|------|-----------------|------|-----------------|------|
| 32  | $4.46 \times 10^{-4}$ | –    | $5.68 \times 10^{-4}$ | –    | $1.28 \times 10^{-3}$ | –    |
| 64  | $1.84 \times 10^{-4}$ | 1.28 | $2.30 \times 10^{-4}$ | 1.31 | $4.73 \times 10^{-4}$ | 1.44 |
| 128 | $5.22 \times 10^{-5}$ | 1.82 | $6.48 \times 10^{-5}$ | 1.83 | $1.31 \times 10^{-4}$ | 1.86 |
| 256 | $1.36 \times 10^{-5}$ | 1.94 | $1.68 \times 10^{-5}$ | 1.95 | $3.37 \times 10^{-5}$ | 1.95 |
| 512 | $3.45 \times 10^{-6}$ | 1.98 | $4.26 \times 10^{-6}$ | 1.98 | $8.55 \times 10^{-6}$ | 1.98 |

TABLE II: Error norms and convergence rates for viscous incompressible flow in the Taylor-Green vortex.
FIG. 4: Poiseuille flow in a cylinder, slices in the $\hat{x}\hat{y}$-plane. The velocity magnitude is shown in (a), while (b) depicts the relative deviation of the stress from the yield stress threshold. In the latter plot, values outside the colormap are mapped on to the endpoints.

The dashed line is $\hat{r} = r_0$.

velocity. Surrounding it, an annulus of constant shear rate leads to the parabolic profile we expect between the wall and the yield surface. The yield surface is characterised by the stress contour $\|\mathbf{\tau}\| = \tau_0$. However, as discussed previously in the literature, there is instability near this stress value which means that a better measure of the fully converged yield surface is the contour $\|\mathbf{\tau}\| = (1 + \delta)\tau_0$, where $\delta$ is some small parameter of the order $10^{-346–48}$. This is because the solution converges much faster in the yielded region than in the unyielded ones. On the other hand, Treskatis argues that a better visual investigation of the yield surface is obtained by plotting the relative deviation from the yield surface, $\|\mathbf{\tau}\| / \tau_0 - 1$, restricted to some small range around zero. In this manner, we avoid the introduction of systematic error through overestimation of the unyielded regions. Note that this is done using a colormap which changes abruptly at zero, as seen in figure 4b.

From figure 4b we can see that there is a sharp transition from the yielded to the unyielded regions which is in agreement with the analytical solution for the yield surface, $\hat{r} = r_0$. This validates that our model accurately captures the Bingham properties of the
FIG. 5: Comparison of our numerical results with the analytical velocity profile for Poiseuille flow of a Bingham fluid in a circular cylinder.

fluid, and that the regularisation parameter is small enough at $\varepsilon = 0.01 \text{s}^{-1}$ for the Poiseuille problem, although a smaller value will be necessary for more demanding cases.

In figure 5, the analytical velocity profile as given by (35) is compared with our computational results at steady-state for three mesh resolutions. It is evident that even for these moderate resolutions, and with the relatively large regularisation parameter $\varepsilon$, the numerical solution collapses on to the analytical one with improved mesh resolution. Furthermore, it is worth noting that the discrepancy from the analytical solution is largest in the unyielded region, whereas the profile is clearly captured in the shear region near the wall.

In order to properly evaluate the capabilities of our code, we need to simulate test cases which exhibit fully three-dimensional effects which are characteristic of yield stress fluids. A widely discussed case is that of bodies moving at constant speed through a Bingham fluid. Such bodies are fully encapsulated by a so-called yield envelope, separating the unyielded bulk material from an interior recirculating flow. This interior flow has a unique topology owing to the viscoplastic nature of the fluid, and has been widely studied for two-dimensional\(^{50,56}\) and axisymmetric three-dimensional\(^{52,58}\) shapes.
C. Cylinder moving through Bingham fluid

Although more works have traditionally focused on the case of a sphere translating in a viscoplastic fluid\cite{Simmans2014, Simmans2016, Simmans2017, Simmans2018a, Simmans2018b}, the simpler case is that of an infinitely long circular cylinder. This is both due to the fact that only a thin slice is required for comparisons with two-dimensional reference results, but also because the viscoplastic effects are more dominant due to the extended geometry. Consequently, there is no ambiguity regarding the resulting shape of yield surfaces. An excellent reference is the paper by Tokpavi et al.\cite{Tokpavi2015}, in which the flow around a highly resolved 2D quadrant surrounding the cylinder is simulated with a Papanastasiou regularisation scheme. They show that the occurrence of characteristic dips in the yield envelope fore and aft of the cylinder is clearly prominent for Bingham fluids, in addition to small unyielded caps attached to the poles of the cylinder and rigidly rotating unyielded plugs in its equatorial plane. The same has been demonstrated by Chaparian and Frigaard\cite{Chaparian2017} by using an augmented Lagrangian approach, which captures the yield surface without regularisation. They also show how slip line field theory captures the yield envelope and polar caps well, but is ill-suited for the equatorial plugs. It is also worth mentioning that there have been a number of studies with multiple cylinders in various colinear configurations, all showing similar large-scale features\cite{Simmans2015, Simmans2016, Simmans2017}. Since the problem is essentially two-dimensional due to its planar symmetry, we expect to see the same yield surface shapes as the aforementioned authors, even though our code is fully three-dimensional.

We thus consider an infinitely long circular cylinder, where the direction of flow (along \( \hat{z} \)) is perpendicular to the cylinder axis (\( \hat{y} \)). The computational domain is \( \Omega = [0, 4D] \times [0, D/4] \times [0, 6D] \), where \( D \) is the diameter of the cylinder, which we take as the characteristic length scale. Rather than imposing a constant velocity on the cylinder, we keep it fixed and instead let \( \mathbf{u} = \mathcal{U} \hat{z} \) everywhere initially and at the domain boundaries as the simulation progresses. This means that the cylinder is falling with the same speed (\( \mathcal{U} \)) in the reference frame where the bulk fluid is at rest. Periodic boundary conditions are imposed in the axial direction of the cylinder.

There are two dimensionless numbers which govern the flow, namely the Reynolds number

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

(38)

(38)
and the Bingham number

\[ Bi = \frac{\tau_0 D}{\mu U}, \quad (39) \]

which quantifies the relative strength of yield stress to viscous stress. We are interested in
the creeping flow regime with a high degree of viscoplasticity, so we let \( Re = 1 \) and take
\( Bi = 340.7 \), the latter simply to match the Bingham number in the study by Liu, Muller and
Denn\(^{53} \), where it is stated that this seemingly arbitrary number is large enough to diminish
any outer boundary wall effects for the given choice of domain size.

Although the reference results only include the location of the yield surfaces, there is a
lot of insight to be gained from visualising other metrics of the flow field. Figure 6 therefore
illuminates the resulting flow field through plots of several different metrics, for the case
with \( N = D/\Delta x = 64 \) and \( \varepsilon = 10^{-3} \text{ s}^{-1} \). The upper left plot, (a), shows the relative speed
with velocity vectors overlaid. A distinct boundary is visible, enclosing a region of non-zero
relative velocity surrounding the cylinder. This boundary represents the yield envelope of
the body. The velocity vectors indicate a recirculating flow, sweeping material away from
the front of the travelling cylinder to the rear in a wide, circular arc. This results in slowly
moving material either side of the cylinder, in addition to material travelling at the same
speed as the cylinder at the polar caps. These observations are indicative of the expected
unyielded plugs rotating at the equator and clinging to the polar caps.

In the upper right plot, (b), the magnitude of the rate-of-strain tensor is shown with
logarithmic scaling for the colormap. This plot properly elucidates the low-strain regions of
unyielded fluid, and confirm the existence of rigidly rotating equatorial plugs and unyielded
material attached to the polar caps, as implied by the velocity distribution. Note that
although the yield surface is characterised by the contour \( \| \dot{\gamma} \| = 0 \) for ideal Bingham plastics
as given by (5), the regularised version (6) leads to a yield surface given by

\[ \| \dot{\gamma} \| = \varepsilon W \left( \frac{\tau_0}{\mu \varepsilon} \right), \quad (40) \]

where \( W(x) \) is the Lambert \( W \) function, also known as the product logarithm since it is
defined as the solution to \( x = W(xe^x) \). With the current parameter values, (40) gives a
yield surface characterised by \( \| \dot{\gamma} \| = 0.010397 \text{ s}^{-1} \), which is why we have used \( 10^{-2} \text{ s}^{-1} \) as
the lower limit of the colormap.

Figure 6c shows the relative stress deviation from the yield surface, just like figure 4b
did for the Poiseuille case. For this more demanding problem, the transition is much less
FIG. 6: Flow characteristics in the \( \hat{x}\hat{z} \)-plane through the centre of the cylinder at steady-state. Depicted are (a) the relative velocity field and its magnitude, (b) the rate-of-strain magnitude, (c) the yield surface stress deviation and (d) the flow topology parameter. Note that the colormap in (b) has logarithmic scaling, and that values outside the colormap for (c) are mapped on to the endpoints. In (d), the yield surface computed with \( \delta = 10^{-3} \) is also masked out in grey. In all plots, the cylinder is masked out in black.
sharp, but all the expected unyielded regions (envelope, rotating plugs and polar caps) are clearly captured where the stress deviation is negative. The actual yield surface is located somewhere in the transition from blue to white.

Finally, we have plotted a flow topology parameter which is a normalised invariant of the velocity gradient tensor. It indicates the relative strengths of the strain-rate and the vorticity\(^60,61\),

\[
\Lambda = \frac{\left\| \dot{\gamma} \right\|^2 - \left\| \omega \right\|^2}{\left\| \dot{\gamma} \right\|^2 + \left\| \omega \right\|^2}.
\]

This parameter accurately describes the nature of different parts of the flow regime, since values of \(\Lambda\) equal to \(-1\), 0 and 1 correspond to pure rotation, shear and extension, respectively. As expected, we observe pure rotation around the unyielded plugs on the equatorial line of the cylinder. Adjacent to areas of rigid rotation, and near the cylinder, shear flow is evident. This is also the case near the yield envelope. In fact, these two shearing regions represent two distinct cases of viscoplastic boundary layers, as discussed by Balmforth in his recently published lecture notes\(^62\). Finally, a belt of purely extensional flow surrounds the cylinder, rotating plugs and polar caps. Note that in this plot, the yield surfaces masked out in grey are computed with \(\delta = 10^{-3}\). This allows us to verify that we recover the same yield surface shapes as those computed in the references\(^54,55\), when using the same visualisation procedure.

D. Sphere moving through Bingham fluid

Since the cylinder test case allows for direct comparisons with two-dimensional simulations, it is a good test case for verifying the interplay between yield stress rheology and the embedded boundaries. On the other hand, it does not exhibit any genuinely three-dimensional effects, and for many real-world scenarios an infinite cylinder is not a realistic representation of the actual bluff bodies. We therefore investigate the flow around a sphere in the same configuration, retaining its diameter \(D\) as the characteristic length scale, but extending the domain to \([0, 4D]\) in the \(y\)-direction. This problem has been analysed by several authors previously\(^50,53,55\), but it still warrants further attention due to gaps relating to important viscoplastic flow features. In particular, claims about how specific parts of the yield surface depend on spatial resolution and regularisation parameter are not consistent. Furthermore, few of them were based on three-dimensional simulations, and those that were
tended to be hampered by low spatial resolution, making it difficult to draw any conclusions regarding the shape and extent of the yield surface.

Ansley and Smith first considered the problem in 1967, and proposed a yield surface based on slip line field theory. This included dips in the yield envelope and polar caps. Although it was a great contribution, they conceded that the qualitative shape of the yield surface could not be supported by direct evidence. Beris et al. phrased the problem as a free boundary problem at the yield surface, and performed 2D simulations with a priori estimates of its location. They captured the envelope dips and polar caps, and reported an equatorial torus-shaped region with low strain-rate. Its motion was described as close to the sum of rigid translation in the direction of flow and solid body rotation, but they aptly noted that perfectly rigid rotation is not possible in the configuration, except exactly in the equatorial plane. This is due to the rotated strain field around the equator of the sphere. Liu et al. computed 2D results in a quadrant using Papanastasiou regularisation and finite elements on a body fitted mesh. They reported the existence of both polar caps and an equatorial torus, although the latter is unphysical according to the arguments put forth by Beris et al. They showed that the plug regions shrank with decreasing regularisation parameter, but could not infer the limiting behaviour. More recently, three-dimensional simulations in the framework of lattice Boltzmann techniques have been published, utilising a dual viscosity model and Papanastasiou regularisation. However, the resolution for these simulations was low \((N = 12)\), so it is difficult to draw accurate conclusions regarding the yield surface shapes.

The two parameters which determine the yield surface for a given Bingham number, are the regularisation parameter \(\varepsilon\) and the spatial resolution as given by \(N\). Similarly to what was done in the paper by Liu et al., in Figure we plot the computed yield surfaces in the upper right corner of the plane \(\hat{y} = 2\), which goes directly through the sphere, for a range of \(\varepsilon\) values. However, we also consider three different mesh resolutions, in order to separate the effects of the two parameters. It is clear that both parameters must have acceptable values in order for the solution to converge as expected. Without a small enough regularisation parameter (at least \(\varepsilon = 10^{-3}\)), the yield envelope dips and low-strain equatorial plugs are not captured, whereas a low spatial resolution results in unresolved polar caps. We note that the unphysical torus-shaped yield surface at the equator is apparent also in our simulations, but this is due to the utilisation of \(\delta = 10^{-3}\) in the visualisation of the yield surface, as
FIG. 7: Contour lines of the steady-state yield surface $\|\tau\| = 1.001 \tau_0$ in the upper right corner of the $\hat{x}\hat{z}$-plane $\hat{y} = 2$. The resolution of the numerical mesh employed in the simulations increases from (a) to (c). For each resolution, the different lines correspond to the following regularisation parameter values:

- $\epsilon = 10^{-1}$
- $\epsilon = 10^{-2}$
- $\epsilon = 10^{-3}$
- $\epsilon = 10^{-4}$

explained below.

In figure 8, we show plots of the same variables as those in figure 6 for the cylinder. There are several noteworthy points to make about the differences between the two cases. Firstly, the overall size of the yield envelope is significantly smaller for the sphere than the cylinder, both in the direction of flow and in the $xy$-plane. Secondly, the unyielded regions within the envelope are much smaller. Both of these observations are as expected, since the axial symmetry of the sphere causes reduced shearing effects in the plane. Equivalently, the extended cylinder enhances shearing in the plane, leading to the larger unyielded regions and genuine rigid body rotation. Finally, the relative stress deviation plot (c) proves that there aren’t any regions in the equatorial torus where the stress is actually below the yield stress threshold. Although the stress gets extremely close to the yield threshold in these areas of low strain, there is no evidence of a genuinely unyielded region, since the relative stress plot stays blue. The diffuse interfaces in this plot shed some light on why the torus-shaped plug is present in some papers and not others: it is severely close to the threshold value without actually reaching it, and without a sharp transition near it. Thus, the benefits of the visualisation strategy introduced by Treskatis becomes clear: the plot in (c) shows in details what the stress deviation looks like near the yield surface, while the well-known masked out contour $\|\tau\| = 1.001 \tau_0$ in (d) oversimplifies the case. This is all in agreement
FIG. 8: Steady-state flow field around the sphere in the plane $\hat{y} = 2$.

with the original arguments put forth by Beris et al. Figure 9 shows the last two plots through the slice $\hat{z} = 2$, i.e. perpendicular to the flow direction, but still through the centre of the sphere.

Finally, in figure 10 we visualise three-dimensional stress contours in the vicinity of the sphere, computed with $\delta = 10^{-3}$. Since the yield envelope fully encloses the sphere and plug regions, its opacity is reduced from subfigures (a) to (c), in order to reveal the shape of the enclosed yield surfaces. Note again that the toroidal yield surface at the equator disappears.
FIG. 9: Steady-state flow field around the sphere in the plane \( \hat{z} = 2 \).

FIG. 10: Yield surface represented by the contour \( \|\tau\| = 1.001\tau_0 \) (light grey) around sphere (magenta) moving at constant speed through a Bingham fluid. The opacity of the yield envelope is reduced from left to right, in order to reveal the sphere and unyielded regions within it. The right-most plot (c) is enlarged compared to the other two.

in the limit \( \delta \to 0 \). This kind of visualisation is entirely novel to the best of our knowledge, and gives a richer picture of the yield surface topology. Since there is no body-fitted mesh or assumption on symmetry in the flow, this type of simulation opens a range of possibilities for investigating flow patterns and yield surfaces in non-trivial configurations.
E. Non-trivial particle shape

As a final demonstration, we replace the sphere by an object which is the union of a sphere and cube, and orient it so that the flow field becomes asymmetrical. This means that a three-dimensional representation is necessary to capture the fluid dynamics. To be precise, the sphere is the same size as in the previous section, but has centre coordinates \((\hat{x}, \hat{y}, \hat{z}) = (1.8, 1.8, 2.8)\), while the cube is defined by two points on opposing sides of its main diagonal, in dimensionless units \((1.85, 1.85, 2.85)\) and \((2.5, 2.5, 3.5)\). The resulting 3D yield surface, computed with \(\delta = 10^{-3}\), is shown in figure 11. In contrast to figure 10, we reduce the opacity of all the yield surfaces, and not just the enclosing envelope. This is due to the difficulty in separating the locations of the yield surface types by simple rules. Although the stress contours are now much more complex, we can still recognise the expected traits: an enclosing yield envelope surrounding the entire body, in addition to smaller unyielded plugs attached to it at places of low strain rate. In particular, there are caps of unyielded material fore and aft of the object in the flow direction, as well as along the narrow intersection of the sphere and cube. Additionally, the characteristic low-strain rotating region around the sphere’s equator is visible, but it does not continue around the entire object. The free cube sides, which are aligned with the flow direction, only lead to a very narrow boundary layer separating the object from the yield envelope. A video is available online, in which the observer’s point of view is rotated around the object in order to show the yield surface from all polar angles.

V. CONCLUSIONS

We have presented a new methodology for simulating highly resolved flow of generalised Newtonian fluids (in particular, viscoplastics) in three-dimensional domains with non-trivial boundaries. The domain geometry is treated using embedded boundaries, which are applied to viscoplastic fluid computations for the first time. Highly resolved simulations are made possible by implementing the algorithm in AMReX, a high performance library of tools for solving PDEs on structured grids. The solver is robust and efficient, and allowed us to perform simulations of various objects moving at constant relative speed through an infinite volume of Bingham fluid, revealing the shape of the resulting yield envelopes and
FIG. 11: Yield surface computed with $\delta = 10^{-3}$ for the flow of a Bingham fluid around the union of a sphere and a cube. The opacity of the stress contours is reduced to 50% in the right plot, in order to show unyielded regions enclosed by the outer yield envelope.

Multimedia view available online.

plug regions. We hope our contribution can prove a useful tool to researchers interested in studying three-dimensional flows of Bingham fluids in or around domains with non-trivial shapes.

ACKNOWLEDGMENTS

K. S. would like to acknowledge the EPSRC Centre for Doctoral Training in Computational Methods for Materials Science for funding under grant number EP/L015552/1. Additionally, he acknowledges the funding and technical support from BP through the BP International Centre for Advanced Materials (BP-ICAM) which made this research possible. Finally, he is grateful to Arndt Ryo Koblitz for useful discussions and for sharing his valuable insights.
Appendix A: Polynomial interpolation

Given a set of \( n+1 \) real values \( y_i \) at data points \( x_i \) for \( i \in \{0, \ldots, n\} \), polynomial interpolation seeks to find a polynomial curve whose curve passes through each of them. The Lagrange form of the interpolant is given by

\[
f(x) = \sum_{i=0}^{n} y_i \prod_{j \neq i} \frac{x - x_j}{x_i - x_j}, \tag{A1}\]

so that it’s derivative is

\[
\frac{\partial f}{\partial x} = \sum_{i=0}^{n} y_i \sum_{j \neq i} \frac{x - x_j}{x_i - x_j} = \sum_{i=0}^{n} y_i \prod_{j \neq i} \frac{x - x_j}{x_i - x_j}. \tag{A2}\]

The polynomial interpolant is unique and accurate to order \( n \).

REFERENCES

1. K. Sverdrup, N. Nikiforakis, and A. Almgren, “Highly parallelisable simulations of time-dependent viscoplastic fluid flow with structured adaptive mesh refinement,” Physics of Fluids 30, 093102 (2018).
2. P. Saramito and A. Wachs, “Progress in numerical simulation of yield stress fluid flows,” Rheologica Acta 56, 211–230 (2017).
3. E. Mitsoulis and J. Tsamopoulos, “Numerical simulations of complex yield-stress fluid flows,” Rheologica Acta 56, 231–258 (2017).
4. A. Koblitz, S. Lovett, and N. Nikiforakis, “Direct numerical simulation of particle sedimentation in a Bingham fluid,” Physical Review Fluids 3, 093302 (2018).
5. N. Kanaris, S. Kassinos, and A. Alexandrou, “On the transition to turbulence of a viscoplastic fluid past a confined cylinder: A numerical study,” International Journal of Heat and Fluid Flow 55, 65–75 (2015).
6. E. Chaparian, A. Wachs, and I. A. Frigaard, “Inline motion and hydrodynamic interaction of 2d particles in a viscoplastic fluid,” Physics of Fluids 30, 033101 (2018).
7. A. Roosing, O. Strickson, and N. Nikiforakis, “Fast distance fields for fluid dynamics mesh generation on graphics hardware,” arXiv preprint arXiv:1903.00353 (2019).
8. A. Wachs, “Computational methods for viscoplastic fluid flows,” in Lectures on Viscoplastic Fluid Mechanics (Springer, 2019) pp. 83–125.
9 J. W. Purvis and J. E. Burkhalter, “Prediction of critical Mach number for store configurations,” AIAA Journal 17, 1170–1177 (1979).
10 B. Wedan and J. South, Jr., “A method for solving the transonic full-potential equation for general configurations,” in 6th Computational Fluid Dynamics Conference Danvers (1983) p. 1889.
11 D. K. Clarke, H. Hassan, and M. Salas, “Euler calculations for multielement airfoils using Cartesian grids,” AIAA journal 24, 353–358 (1986).
12 S. Falle and J. Giddings, “An adaptive multigrid applied to supersonic blunt body flow,” Numerical Methods in Fluid Dynamics, III (1988).
13 M. Berger and R. Leveque, “An adaptive Cartesian mesh algorithm for the Euler equations in arbitrary geometries,” in 9th Computational Fluid Dynamics Conference (1989) p. 1930.
14 P. Colella, D. T. Graves, B. J. Keen, and D. Modiano, “A Cartesian grid embedded boundary method for hyperbolic conservation laws,” Journal of Computational Physics 211, 347–366 (2006).
15 H. Johansen and P. Colella, “A Cartesian grid embedded boundary method for Poisson’s equation on irregular domains,” Journal of Computational Physics 147, 60–85 (1998).
16 P. McCorquodale, P. Colella, and H. Johansen, “A Cartesian grid embedded boundary method for the heat equation on irregular domains,” Journal of Computational Physics 173, 620–635 (2001).
17 M. Berger, “Cut cells: meshes and solvers,” in Handbook of Numerical Analysis, Vol. 18 (Elsevier, 2017) pp. 1–22.
18 N. Gokhale, N. Nikiforakis, and R. Klein, “A dimensionally split Cartesian cut cell method for hyperbolic conservation laws,” Journal of Computational Physics 364, 186–208 (2018).
19 N. Gokhale, N. Nikiforakis, and R. Klein, “A dimensionally split Cartesian cut cell method for the compressible Navier–Stokes equations,” Journal of Computational Physics 375, 1205–1219 (2018).
20 H. A. Barnes, J. F. Hutton, and K. Walters, An introduction to rheology, Vol. 3 (Elsevier, 1989).
21 J. Mewis, “Thixotropy – a general review,” Journal of Non-Newtonian Fluid Mechanics 6, 1–20 (1979).
22 H. A. Barnes, “Thixotropy – a review,” Journal of Non-Newtonian Fluid Mechanics 70, 1–33 (1997).
23T. C. Papanastasiou, “Flows of materials with yield,” Journal of Rheology 31, 385–404 (1987).

24P. de Souza Mendes and E. S. Dutra, “Viscosity function for yield-stress liquids,” Appl. Rheol 14, 296–302 (2004).

25A. De Waele, “Die Änderung der Viskosität mit der Scheergeschwindigkeit disperser Systeme,” Colloid & Polymer Science 36, 332–333 (1925).

26E. C. Bingham, “An investigation of the laws of plastic flow,” Bulletin of the Bureau of Standards 13, 309–353 (1916).

27J. Oldroyd, “A rational formulation of the equations of plastic flow for a Bingham solid,” in Mathematical Proceedings of the Cambridge Philosophical Society, Vol. 43 (Cambridge University Press, 1947) pp. 100–105.

28W. H. Herschel and R. Bulkley, “Konsistenzmessungen von Gummi-benzollösungen,” Colloid & Polymer Science 39, 291–300 (1926).

29A. Almgren, J. Bell, P. Colella, L. Howell, and M. Welcome, “A conservative adaptive projection method for the variable density incompressible Navier–Stokes equations,” Journal of Computational Physics 142, 1–46 (1998).

30R. B. Pember, L. H. Howell, J. B. Bell, P. Colella, W. Y. Crutchfield, W. Fiveland, and J. Jessee, “An adaptive projection method for unsteady, low-Mach number combustion,” Combustion Science and Technology 140, 123–168 (1998).

31M. S. Day and J. B. Bell, “Numerical simulation of laminar reacting flows with complex chemistry,” Combustion Theory and Modelling 4, 535–556 (2000).

32A. Almgren, V. Beckner, J. Bell, M. Day, L. Howell, C. Joggerst, M. Lijewski, A. Nonaka, M. Singer, and M. Zingale, “CASTRO: A new compressible astrophysical solver. I. Hydrodynamics and self-gravity,” The Astrophysical Journal 715, 1221 (2010).

33A. S. Almgren, J. B. Bell, M. J. Lijewski, Z. Lukić, and E. Van Andel, “Nyx: A massively parallel AMR code for computational cosmology,” The Astrophysical Journal 765, 39 (2013).

34A. Dubey, A. Almgren, J. Bell, M. Berzins, S. Brandt, G. Bryan, P. Colella, D. Graves, M. Lijewski, F. Löffler, B. O’Shea, E. Schnetter, B. Van Straalen, and K. Weide, “A survey of high level frameworks in block-structured adaptive mesh refinement packages,” Journal of Parallel and Distributed Computing 74, 3217–3227 (2014).
35. S. Habib, A. Pope, H. Finkel, N. Frontiere, K. Heitmann, D. Daniel, P. Fasel, V. Morozov, G. Zagaris, T. Peterka, V. Vishwanath, Z. Lukić, S. Sehrish, and W.-k. Liao, “HACC: Simulating sky surveys on state-of-the-art supercomputing architectures,” New Astronomy 42, 49–65 (2016).

36. P. Wesseling and C. W. Oosterlee, “Geometric multigrid with applications to computational fluid dynamics,” Journal of Computational and Applied Mathematics 128, 311–334 (2001).

37. H. A. Van der Vorst, “Bi-CGSTAB: A fast and smoothly converging variant of Bi-CG for the solution of nonsymmetric linear systems,” SIAM Journal on Scientific and Statistical Computing 13, 631–644 (1992).

38. https://github.com/AMReX-Codes/incflo

39. https://amrex-codes.github.io

40. M. Kang, R. P. Fedkiw, and X.-D. Liu, “A boundary condition capturing method for multiphase incompressible flow,” Journal of Scientific Computing 15, 323–360 (2000).

41. F. H. Harlow and J. E. Welch, “Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface,” Physics of Fluids 8, 2182–2189 (1965).

42. A. S. Almgren, J. B. Bell, and W. Y. Crutchfield, “Approximate projection methods: Part I. inviscid analysis,” SIAM Journal on Scientific Computing 22, 1139–1159 (2000).

43. D. Graves, P. Colella, D. Modiano, J. Johnson, B. Sjogreen, and X. Gao, “A Cartesian grid embedded boundary method for the compressible Navier–Stokes equations,” Communications in Applied Mathematics and Computational Science 8, 99–122 (2013).

44. R. J. LeVeque, “High resolution finite volume methods on arbitrary grids via wave propagation,” Journal of Computational Physics 78, 36–63 (1988).

45. R. J. LeVeque, “Cartesian grid methods for flow in irregular regions,” Numerical Methods in Fluid Dynamics 3, 375–382 (1988).

46. Z. Yu and A. Wachs, “A fictitious domain method for dynamic simulation of particle sedimentation in Bingham fluids,” Journal of Non-Newtonian Fluid Mechanics 145, 78–91 (2007).

47. M. A. Olshanskii, “Analysis of semi-staggered finite-difference method with application to Bingham flows,” Computer Methods in Applied Mechanics and Engineering 198, 975–985 (2009).
48. J. C. De los Reyes and S. G. Andrade, “Numerical simulation of two-dimensional Bingham fluid flow by semismooth Newton methods,” Journal of Computational and Applied Mathematics 235, 11–32 (2010).

49. T. Treskatis, M. A. Moyers-Gonzalez, and C. J. Price, “An accelerated dual proximal gradient method for applications in viscoplasticity,” Journal of Non-Newtonian Fluid Mechanics 238, 115–130 (2016).

50. R. W. Ansley and T. N. Smith, “Motion of spherical particles in a Bingham plastic,” AIChE Journal 13, 1193–1196 (1967).

51. A. Beris, J. Tsamopoulos, R. Armstrong, and R. Brown, “Creeping motion of a sphere through a Bingham plastic,” Journal of Fluid Mechanics 158, 219–244 (1985).

52. J. Blackery and E. Mitsoulis, “Creeping motion of a sphere in tubes filled with a Bingham plastic material,” Journal of Non-Newtonian Fluid Mechanics 70, 59–77 (1997).

53. B. T. Liu, S. J. Muller, and M. M. Denn, “Convergence of a regularization method for creeping flow of a Bingham material about a rigid sphere,” Journal of Non-Newtonian Fluid Mechanics 102, 179–191 (2002).

54. D. L. Tokpavi, A. Magnin, and P. Jay, “Very slow flow of Bingham viscoplastic fluid around a circular cylinder,” Journal of Non-Newtonian Fluid Mechanics 154, 65–76 (2008).

55. E. Chaparian and I. A. Frigaard, “Yield limit analysis of particle motion in a yield-stress fluid,” Journal of Fluid Mechanics 819, 311–351 (2017).

56. A. R. Koblitz, S. Lovett, and N. Nikiforakis, “Viscoplastic squeeze flow between two identical infinite circular cylinders,” Physical Review Fluids 3, 023301 (2018).

57. Prashant and J. Derksen, “Direct simulations of spherical particle motion in Bingham liquids,” Computers & Chemical Engineering 35, 1200–1214 (2011).

58. S.-G. Chen, C.-H. Zhang, Y.-T. Feng, Q.-C. Sun, and F. Jin, “Three-dimensional simulations of Bingham plastic flows with the multiple-relaxation-time lattice Boltzmann model,” Engineering Applications of Computational Fluid Mechanics 10, 346–358 (2016).

59. B. T. Liu, S. J. Muller, and M. M. Denn, “Interactions of two rigid spheres translating collinearly in creeping flow in a Bingham material,” Journal of Non-Newtonian Fluid Mechanics 113, 49–67 (2003).

60. P. Davidson, Turbulence: an introduction for scientists and engineers (Oxford University Press, 2015).
S. De, J. Kuipers, E. Peters, and J. Padding, “Viscoelastic flow simulations in model porous media,” Physical Review Fluids 2, 053303 (2017).

N. J. Balmforth, “Viscoplastic asymptotics and other analytical methods,” in Lectures on Visco-Plastic Fluid Mechanics (Springer, 2019) pp. 41–82.