AN EXTENDED FINITE ELEMENT METHOD BASED APPROACH FOR LARGE DEFORMATION FLUID-STRUCTURE INTERACTION

Axel Gerstenberger and Wolfgang A. Wall

Technische Universität München, Fakultät Maschinenwesen
Boltzmannstr. 15, 85747 Garching, Germany
e-mail: gerstenberger@lnm.mw.tum.de
web page: http://www.lnm.mw.tum.de/Members/gerstenberger

Key words: fluid-structure interaction, extended finite element method, fixed grid, fictitious domain method, immersed boundary

Abstract. This paper illustrates aspects of an ongoing effort to develop a fixed grid fluid-structure interaction scheme that can be applied to the interaction of most general structures with incompressible flow. After presenting a list of requirements for future fixed grid methods, an eXtended Finite Element Method (XFEM) based fixed grid method is proposed. It will allow the simulation of large deformations of thin and bulky structures. The extended Eulerian fluid field and the Lagrangian structural field are coupled using an partitioned, iterative approach. Finally, first results illustrating the essential capabilities are presented.

1 Introduction

Fluid-structure interaction is of great relevance in many fields of engineering as well as in the applied sciences. Hence the development and application of respective simulation approaches has gained great attention over the past decades. Some current endeavors in this field are: the advancement from special purpose or special problem to quite general approaches; the desire to even capture very general and complex systems; and the exigent need of robust high quality approaches even for such complex cases, i.e. approaches that have the potential to turn over from being a challenging and fascinating research topic to a development tool with real predictive capabilities [1]. Often, when interaction effects are essential this comes along with large structural deformations. However, many available approaches (both in research as well as in commercial codes) lack robustness especially in this situation.

A sketch of the general problem of the interaction of a flow field and a flexible structure, in this case an embedded structure, is shown in Figure 1. The conjoined interface $\Gamma_{fsi}$ separates the structural domain $\Omega$ from the fluid domain $\Omega^f$. Most research and commercial codes that are available for simulations of the interaction of flows and flexible, often
thin-walled, structures are based on the Arbitrary Lagrangian Eulerian (ALE) method. These approaches go back to early works like [2, 3, 4, 5, 6, 7]. The essential feature of ALE based methods is that the fluid field is formulated and solved on a deforming grid. This grid deforms with the structure at the interface and then the grid deformation is extended into the fluid field.

But even the most advanced and best cultured ALE based scheme once comes to its limits where only re-meshing helps. At the latest in such situations, one might be tempted to turn over to approaches that work with a fixed grid. Here, the interface is described either explicitly, using some kind of Lagrangian interface markers or a Lagrangian structural discretization, or implicitly, using e.g. level-set functions on a fixed fluid grid. Changing properties and discontinuities in the fluid solution have to be taken care of with modifications on the fluid equations and/or fluid discretization.

Prominent fixed grid methods for incompressible flow include the Immersed Boundary (IB) method [8, 9] and its many derivations [10, 11, 12, 9]. It is capable to simulate thin and deformable boundaries and fully fledged, deformable 3d structures submerged in incompressible flow [13, 14]. An approach with many similarities to the IB method is the so called Distributed Lagrange Multiplier / Fictitious Domain (DLM/FD) method [15, 16]. Originally, the approach was developed for rigid particles with translational and rotational degrees of freedom. The DLM/FD methods have since been extended to simulate thin, deformable structural surfaces [17, 18, 19] as well as to flexible and fully fledged structures [20]. Both methods have in common a Lagrangian mesh for the structure moving on top of the fluid mesh and forcing the fluid material to deform as the structure. Other fixed-grid or Cartesian-grid methods are, among others, Fedkiw et al. [21] and Cirak & Radovitzky [22].

Without being complete, below we give a list of short-comings of which at least one applies to each of the existing methods and for which we need improvements before fixed
grid methods can become as widespread as ALE based FSI algorithms.

- **Representation of thin and volume occupying structures** Some methods are only available for structures represented as beam/shell models. A widely applicable method has to be able to represent structures as shells and full solids occupying space.

- **Removing the influence of the ‘fictitious’ fluid on the physics and the simulation process** Volume occupying solids create a ‘fictitious’ space ‘within’ the structural domain that should influence the simulation result. This includes several important points such as
  - *Removing the incompressibility constraint from the structural domain* If all Lagrangian points are ‘tied’ to the fictitious fluid by the IB interpolation or a Lagrangian multiplier technique, the structural deformation has to be divergence free \[20, 14\] as well. While this might not be a limitation for applications in biomechanics, where usually water-filled and therefore incompressible tissues are considered, it certainly is a limitation in many engineering applications, since most materials and structures deform in a compressible way. Allowing the structure to deform in a compressible way seems crucial to make fixed grid methods generally applicable.
  - *Artificial viscosity* Despite of the incompressibility problem, the artificial fluid viscosity that is present in the discussed methods for deformable structures has to be removed. For long-time simulation, even a small viscosity will have an impact on the accuracy of the method.
  - *Removing fluid degrees of freedom from within of the structural domain* From an efficiency point of view, it would be advisable to remove the extra fluid unknowns within the structural domain as far as possible. If the structure occupies a significant space of the computational domain, the computational cost of the redundant calculations cannot be ignored.
  - *Empty structures* Removing fluid degrees of freedom from inside the solid can increase the flexibility of the method for another reason. Looking at potential applications, there might be the wish to simulate *e.g.* a shell structure separating two fluids, where one fluid is not simulated at all. Current Fixed Grid methods have no straightforward way for solving such problems.

- **Accurate representation of the kinematic/stress discontinuity** Fixed grid methods can only become serious tools, if the interface and the transfer of variables is represented accurately and consistently as it is possible in many ALE based methods.

- **Mesh size independence** Methods like the IB methods require a certain ratio between fluid and solid mesh size at the interface to simulate impermeable structural surfaces.
Future methods should allow an independent mesh size for each of the simulated fields. Also, there should be no limitation on how thin structures can be with respect to the fluid field grid size.

Until recently, none (to the knowledge of the authors) of the currently available fixed-grid methods meet the high requirements stated above. Among others, these requirements have to be fulfilled in order to match or surpass the ALE approach with respect to applicability, accuracy and numerical stability. They serve us as a guideline for the development of a new fixed-grid approach that promises to overcome the addressed shortcomings.

In an attempt to meet these requirements, we propose an iterative coupling scheme between a standard Lagrangian structural description and an Eulerian formulation for the fluid that uses features of the eXtended Finite Element Method (XFEM) and the DLM/FD methods mentioned above. The XFEM is used to properly describe the interface including discontinuities of the kinematic variables as well as in the momentum balance. The first implementation of such an XFEM based approach for compressible flows was published in Legay et al. [23]. The XFEM was originally introduced for the simulation of Cracks and other discontinuities in structures [24] [25] and have been, close to the topic at hand, extended to problems of two-phase flow [26] and Stokes flow/rigid particle interaction [27]. In this paper, we adopt this scheme and show how the monolithic algorithm in [23] can be modified for an iterative coupling and how bulky and very thin structures can be easily simulated. Furthermore, the use of additional level-sets to describe the interface as used in [23] could be avoided.

With this XFEM based approach, in principal all of the mentioned shortcoming can be addressed, most prominently, there is no influence of the fictitious fluid domain anymore and, for a sufficient large fictitious fluid domain, a significant number of unnecessary fluid unknowns can be removed. The interface can represent the proper discontinuities, although the accuracy and the mesh size dependency between fluid and structural discretization needs further studies.

The paper is structured as follows: After stating the general problem in Section 2, Section 3 covers the addressed coupling issues, while the fluid discretization is given in Section 4. Finally, first preliminary results on the extended fluid parts are shown in Section 5.

2 General fluid-structure interaction problem

A general fluid-structure interaction problem consists of the description of the fluid and solid fields, appropriate fluid-structure interface conditions at the conjoined interface and conditions for the remaining boundaries, respectively. Furthermore, a brief sketch of the solution procedure for each of the fields is presented.
2.1 Fluid

Without consideration of the specific reference system, the conservation of momentum is stated as

\[ \rho^f \frac{Du}{Dt} = \nabla \cdot \sigma + \rho^f b \] (1)

Here, the material time derivative of the velocity \( u \) times the fluid density \( \rho^f \) is balanced by the gradient of the Cauchy stress tensor \( \sigma \) and external, velocity independent volumetric forces \( b \). Mass conservation for an incompressible fluid is stated as

\[ \nabla \cdot u = 0 \] (2)

We use the Newtonian material law, which defines the internal stress tensor \( \sigma \) as

\[ \sigma = -pI + 2\mu \varepsilon(u) \] (3)

It is composed from the pressure \( p \) and a product of the strain rate tensor \( \varepsilon \) given by

\[ \varepsilon(u) = \nabla u + (\nabla u)^T \] (4)

and the dynamic viscosity denoted as \( \mu \).

The material time derivative depends on the choice of the reference system, which itself depends on the way the moving boundary is treated. There are basically three alternative reference systems: the Eulerian, the Lagrangian and the Arbitrary Lagrangian Eulerian (ALE) formulation. For the fixed grid method proposed in this work, the Eulerian formulation is used, where the momentum equation, with the incompressibility constrain inserted, reads as

\[ \frac{\partial u}{\partial t} = -u \cdot \nabla u - \frac{1}{\rho^f} \nabla p + 2\nu \nabla \cdot \varepsilon(u) + b \] (5)

The dynamic viscosity \( \mu \) was replaced by the kinematic viscosity \( \nu = \mu/\rho^f \)

2.2 Structure

Most commonly, the structure is described using a Lagrangian description, where the material derivative becomes a partial derivative with respect to time, such that

\[ \rho^s \frac{\partial^2 d}{\partial t^2} = \nabla \cdot \sigma + \rho^s b \] (6)

with the displacement \( d \) defined as the difference between the current position \( x \) and the initial position \( X \).

In the large deformation case it is common to describe the constitutive equation using a stress-strain relation based on the Green-Lagrange strain tensor \( E \) and the 2. Piola-Kirchhoff stress tensor \( S(E) \) as a function of \( E \). The 2. Piola-Kirchhoff stress can be obtained from the Cauchy stress \( \sigma \) as

\[ S = JF^{-1} \cdot \sigma \cdot F^{-T} \] (7)
and the Green-Lagrange strain tensor \( E \) as

\[
E = \frac{1}{2}(F^T F - I)
\]  

(8)

\( J \) denotes the determinant of the deformation gradient tensor \( F \), which itself is defined as

\[
F = \frac{\partial x}{\partial X}
\]

(9)

For the numerical results presented in this article we use the St.-Venant-Kirchhoff material law for simplicity.

2.3 Boundary and Interface Conditions

The main conditions at the interface are the dynamic and kinematic coupling conditions. The force equilibrium requires the stress vectors to be equal as

\[
\sigma^f \cdot n = \sigma^s \cdot n \quad \forall x \in \Gamma_{fsi}
\]

(10)

For abbreviation, the fluid surface forces at the interface are also referred to as \( t^f \equiv \sigma^f \cdot n \) and \( t^s = \sigma^s \cdot n \).

We assume no mass flow across the interface. Consequently, the normal velocities at the interface have to match as

\[
u \cdot n = \frac{\partial d}{\partial t} \cdot n \quad \forall x \in \Gamma_{fsi}
\]

(11)

If viscous fluids are considered, there is also a matching condition for the tangential velocities, which can be combined with the equation above to obtain the ‘no slip’ boundary conditions as

\[
u = \frac{\partial d}{\partial t} \quad \forall x \in \Gamma_{fsi}
\]

(12)

The remaining boundary conditions can be from Dirichlet or Neumann type and are defined and applied as usual. They will be omitted here for brevity.

2.4 Iterative Staggered Coupling Scheme

The algorithmic implementation of the coupled problem of fluid-structure interaction is based on a field-wise partitioned solution approach. This modular software concept allows for complex and specifically designed approaches for the single fields of the structure and the fluid.

The wet structural surface is the natural interface or coupling surface \( \Gamma_{fsi} \). Complete kinematic and dynamic continuity at \( \Gamma_{fsi} \) would ensure conservation of mass, momentum and energy at the interface. The boundary conditions in this Dirichlet-Neumann substructuring scheme are chosen such that forces generated from the fluid pressure and
viscous friction are exerted on the structural interface as Neumann boundary conditions. The structural displacement at \( \Gamma_{\text{fsi}} \) is transferred into velocities and used as a Dirichlet condition for the fluid field.

For the dynamic FSI problems involving incompressible fluid flow and lightweight structures considered here, we employ an iterative staggered scheme based on [28, 29, 30, 31, 32], where each field is solved implicitly and an iterative procedure over the fields ensures convergence for the interface conditions at the new time step level \( n + 1 \). The algorithm can be summarized as follows:

- Compute an explicit predictor of the structural interface displacement at the new time level \( d_{\Gamma_{\text{fsi}},0}^{n+1} \).

**Fluid:**

- Compute fluid velocity at \( \Gamma_{\text{fsi}} \) from surface position to serve as Dirichlet boundary condition \( u_{\Gamma_{\text{fsi}},i+1}^{n+1} = (d_{\Gamma_{\text{fsi}},i+1}^{n+1}) \).
- Solve fluid equations to obtain \( u_{i+1}^{n+1} \) and \( p_{i+1}^{n+1} \).
- Obtain fluid boundary traction \( t_{f,\Gamma_{\text{fsi}},i+1}^{n+1} \) along \( \Gamma_{\text{fsi}} \).

**Structure:**

- Solve the structural field for the new displacements \( d_{i+1}^{n+1} \) under consideration of the fluid load \( t_{f,\Gamma_{\text{fsi}},i+1}^{n+1} \).
- Relaxation of the interface displacements \( d_{\Gamma_{\text{fsi}},i+1}^{n+1} \) by using \( d_{\Gamma_{\text{fsi}},i+1}^{n+1} = \omega_i d_{\Gamma_{\text{fsi}},i+1}^{n+1} + (1 - \omega_i) d_{\Gamma_{\text{fsi}},i+1}^{n+1} \).
- Check convergence,
- Restart iteration with new fluid field calculation, if not yet converged.

We use Aitken’s acceleration scheme for vector sequences (Irons et al. [33]) to obtain the relaxation parameter \( \omega_i \).

### 3 Moving Fluid Boundaries on Fixed Eulerian grids

Motivated by introductory listed drawbacks of current fixed-grid methods, most prominently the need of removing the influence of the ‘fictitious’ fluid on the structures movement, we propose an approach that uses the properties of XFEM to physically decouple the outside (physical) and inside (fictitious) fluid domain with respect to the structure. As mentioned before, it is particularly helpful to use a partitioned FSI approach, since it allows to state the fluid problem without thinking about the influence of the structures solution process. As can be seen from the algorithmic description of the iterative staggered coupling in the previous section, the fluid-structure interface velocity and the
interfaces position is calculated from the updated structural position. Consequently, the fluid surface position and its velocity are always known at the beginning of a fluid field calculation step. The treatment of the moving fluid-structure interface on a fixed Eulerian grid with the XFEM is presented below.

### 3.1 Re-Stating the Fluid Problem

The principle setup of the proposed approach is depicted in Figure 2. The entire computational domain $\bar{\Omega}$ is defined from the interior domain $\Omega$, the Dirichlet boundaries $\Gamma_{\text{Dirichlet}}$, the Neumann boundaries $\Gamma_{\text{Neumann}}$ and the fluid-structure interface $\Gamma_{\text{fsi}}$ as

$$
\bar{\Omega} = \Omega \cup \Gamma_{\text{Dirichlet}} \cup \Gamma_{\text{Neumann}} \cup \Gamma_{\text{fsi}}
$$

(13)

The interface $\Gamma_{\text{fsi}}$ is defined as the mapping of the structural surface on the fluid grid that is usually not coincident with any fluid element boundaries. The position of the interface is always explicitly known through the structures position and no additional fields like the level-set, as used in crack problems and 2-phase flow problems, are needed. For easier reference, the ‘physical’ fluid domain outside the structure is named $\Omega^+$, the ‘fictitious’ fluid domain inside the structure is named $\Omega^-$. Similarly, we define two additional names for the projected boundary $\Gamma_{\text{fsi}}$, namely $\Gamma^+$ and $\Gamma^-$, depending whether functions are evaluated approaching $\Gamma_{\text{fsi}}$ from $\Omega^+$ or $\Omega^-$, respectively.

The boundary conditions of the FSI problem translate into the following boundary conditions for $\Omega^+$

$$
\mathbf{u} = \bar{\mathbf{u}}_{\text{fsi}} \quad \forall \mathbf{x} \in \Gamma_{\text{fsi}} = \Gamma^+
$$

(14a)
The structural interface velocity serves as a Dirichlet boundary condition for the fluid at $\Gamma^+$, while to the remaining boundaries standard Dirichlet and Neumann conditions can be applied. For $\Omega^-$, the entire boundary is a Neumann boundary with zero traction applied as

$$\mathbf{\sigma}^f \cdot \mathbf{n} = 0 \quad \forall \mathbf{x} \in \Gamma^-$$

The initial conditions are given as

$$u(x,t=0) = u^0 \quad \forall x \in \Omega^+$$

$$p(x,t=0) = p^0 \quad \forall x \in \Omega^+$$

$$u^0 = 0 \quad \forall x \in \Omega^-$$

$$p^0 = 0 \quad \forall x \in \Omega^-$$

Body forces, if present, are only applied to $\Omega^+$, consequently, there will be zero velocity and pressure in $\Omega^-$ at all times, if the initial conditions are set to zero appropriately in $\Omega^-$. 

### 3.2 Discretization in Time

First discretizing the Navier-Stokes equations in time exactly defines the time level at which the position of the interface has to be evaluated and, eventually, how the shape functions have to be constructed.

For simplicity, the one-step-$\theta$ time discretization is employed due to its relative ease of implementation. In the one-step-$\theta$ method, the acceleration $u, t$ is defined as a combination of the new and the old time steps acceleration.

$$\frac{u^{n+1} - u^n}{\Delta t} = \theta [u_i]^{n+1} + (1 - \theta)[u_i]^n$$

Sorting for expressions at the new time step $n + 1$ and the old time step $n$ and replacing $[u_i]^{n+1}$ with the right side of Eq. (5), we yield

$$u^{n+1} - \Delta t\theta[-u \cdot \nabla u - \nabla p + 2\nu \nabla \cdot \varepsilon(u) + b]^{n+1} = u^n + \Delta t(1 - \theta)u_{i,t}$$

$$\nabla \cdot u^{n+1} = 0$$

### 3.3 Weak Form of the Time-Discrete Strong Form

The weak form is developed by multiplying Eq. (18) with the velocity and pressure test functions $v$ and $q$, respectively. The fluid-structure interface condition from Eq. (14) is employed by using a Lagrange multiplier function $\lambda(x)$ along the interface as

$$\lambda \cdot (u - \bar{u}^{\text{fsi}})$$
Axel Gerstenberger and Wolfgang A. Wall

with its test function $\delta \lambda$. The resulting weak form, after integration by parts of the internal stress terms, reads as

$$
\left[(v,u)_{\Omega} + \Delta t \theta ((v,u \cdot \nabla u - b)_{\Omega} - (\nabla \cdot v, p I - 2 \nu \varepsilon(u))_{\Omega} + (\tau^M \nabla q, \nabla p)_{\Omega} - (q, \nabla \cdot u)_{\Omega}) - (\delta \lambda, u)_{\Gamma} + \delta \lambda, \bar{u}_{fsi}^{n+1}\right]_{\Gamma}^{n+1} = (v, u^n)_{\Omega} + \Delta t (1 - \theta) (v, u^n_n)_{\Omega} - (\delta \lambda, \bar{u}_{fsi}^{n+1})_{\Gamma} + (\tau^M \nabla q, \nabla p)_{\Omega} - (q, \nabla \cdot u)_{\Omega} \quad (20)
$$

The Dirichlet and Neumann conditions for the momentum equation are employed as usual, i.e. $v = 0 \forall x \in \Gamma^{\text{Dirichlet}}$ and $\sigma^f \cdot n = \bar{h}^f \forall x \in \Gamma^{\text{Neumann}}$.

Note that only one additional stabilization term, namely $(\tau^M \nabla q, \nabla p)_{\Omega}$, was included, which represents the classical PSPG \cite{34} term with the viscous part omitted. This is the minimal stabilization needed for linear finite elements to prevent pressure oscillation due to the use of equal order Ansatz functions for velocity and pressure. For demonstration purpose and for small Reynolds numbers, this stabilization is sufficient to achieve stable results. The stabilization parameter $\tau^M$ is calculated as in \cite{35} or \cite{36}, while the necessary element size $h_k$ is evaluated at each integration point using the approximate streamlength approach.

### 3.4 Fluid Discretization in Space

#### 3.4.1 Velocity and Stress Discontinuity Across the Fluid-Structure Interface

In the proposed fixed grid scheme, the fluid-structure interface is generally not aligned with fluid element edges. Consequently, the velocity and pressure discontinuity is represented by enhancing the fluid solution space with additional discrete functions using the ideas of XFEM. For example, the velocity is approximated by

$$
u^h(x, t) = \sum_I N_I(x) \left( \bar{u}_I + \psi(x, t) \hat{u}_I \right) \quad (21)
$$

Here, $\bar{u}_I$ represent the standard nodal degrees of freedom at node $I$, while additional degrees of freedom $\hat{u}_I$ multiplied by a properly chosen enrichment function $\psi(x, t)$ are used to enhance the solution with known solutions.

For the fluid-structure interface, a step function $H(x, t)$ defined as

$$
H(x, t) = \begin{cases} 
+1 & \forall x \in \Omega^+ \\
0 & \forall x \in \Omega^- 
\end{cases} \quad (22)
$$

is used as enrichment function $\psi(x, t)$. This particular enrichment function was introduced to represent holes in structures \cite{37}. Likewise, we use this function to represent a ‘hole’ in the fluid. Along $\Gamma^{\text{fsi}}$, both the velocity and the pressure are discontinuous and enriched.
with $H$. The complete discretization for the trial and test functions with equal order Ansatz functions for velocity and pressure are given as

$$\begin{align*}
    u^{h,n+1}(x) & = \sum_I N_I(x)\left(\hat{u}^n_I + H(x, t^{n+1})\hat{u}^{n+1}_I\right) \\
    v^h(x) & = \sum_I N_I(x)\left(\hat{v}_I + H(x, t^{n+1})\hat{v}_I\right)
\end{align*}$$

(23)

and

$$\begin{align*}
    p^{h,n+1}(x) & = \sum_I N_I(x)\left(\hat{p}^n_I + H(x, t^{n+1})\hat{p}^{n+1}_I\right) \\
    q^h(x) & = \sum_I N_I(x)\left(\hat{q}_I + H(x, t^{n+1})\hat{q}_I\right)
\end{align*}$$

(25)

The superscript ‘i’ indicates the discretized field function.

If more than one interface crosses one fluid element, also more enrichments have to be used, using one enrichment for each interface. The additional unknowns $\hat{u}_I^1, \hat{u}_I^2, \ldots$ and enrichment functions $H^1, H^2, \ldots$ are summed as

$$u^h(x) = \sum_I N_I(x)\left(\hat{u}_I + H^1(x)\hat{u}_I^1 + H^2(x)\hat{u}_I^2 + \ldots\right)$$

(27)

with the time dependence omitted for brevity. From an implementation point of view, we will refer to the standard degrees of freedom $\tilde{u}$ also as the zero-th enrichment ($\psi^0 = 1$) as it is a set of nodal unknowns as any of the additional enrichment functions.

As time evolves and the interface moves, the enriched shape functions $N_I(x)\psi(x, t)$ at $n$ and $n + 1$ are generally different near the interface. The right hand side of Eq. (20) shows products of old time step values with the test functions that are defined to test the new solution. Consequently, the solution from the previous time step has to be mapped from the old to the new discretization before Eq. (20) can be solved. Réthoré et al. [38] show an energy conserving mapping between the old and the new time steps configuration that basically consists of proper initialization of new unknowns. The complete mapping will be topic of an upcoming paper.

The Lagrange multiplier field along the boundary is discretized using linear, piecewise continuous shape functions as in [23].

3.4.2 Numerical Integration of the Domain Integrals

Elements divided by discontinuities are subdivided into subdomains using a Delaunay algorithm and integration is performed over sub-domains [25]. This integrates the piecewise linear functions exact, but requires higher programming effort than using higher order Gauss integration schemes.
3.5 Enrichment Strategies

Along the structural surface we distinguish between two possible scenarios: In the first scenario, a segment of the interface, possibly containing several structural element edges, is relatively smooth and without kinks. For such a segment, a continuous enrichment is used as depicted in Figure 3. The Lagrange multiplier mesh is constructed as in Legay et al. [23] using the intersection between the surface and the fluid element boundaries as Lagrange element end nodes.

Note, as we stated before, the velocity and pressure in $\Omega^-$ will remain zero at all times, since no forces apply to its boundaries or its interior. For instance, in an intersected element in $\Omega^-$ the velocity is summed up as

$$u^h(x, t) = 0 = \sum_i N_i(x) (\tilde{u}_i + 0 \hat{u}_i)$$  \hspace{1cm} (28)

Therefore, the standard degree of freedom $\tilde{u}$ will always be zero. This allows to remove these degrees of freedom from intersected elements as shown in Figure 3(b). In addition, all nodal velocities within $\Omega^-$ away from the surface will evaluate to zero and we can safely remove these interior elements. Consequently, we only have to integrate full elements in $\Omega^+$ and subelements that are in $\Omega^+$. Approaches, where integration is performed only on parts of an element to account for discontinuities can also be found in [39] [40] [41].

A thin structure as depicted in Figure 4(a) consists of 2 overlapping smooth enrichments. Again, integrating within $\Omega^-$ has to result in $u^h = 0$ and, since all jump enrichments are zero within $\Omega^-$, the standard degrees of freedom have to be zero and can be removed. With this arrangement, arbitrary thin structures can be represented.
Axel Gerstenberger and Wolfgang A. Wall

The second scenario includes kinks and sharp corners, i.e. the angle between two consecutive segments exceeds a certain value. In this scenario, a kink enrichment is applied as shown in Figure 4(b). Currently, the standard jump enrichment from Eq. (22) is used. Inspired by the crack tip enrichment used in fracture mechanics [24, 37], ideas to improve the tip enrichment will be topic of an upcoming paper.

4 Numerical Examples

In the following section, simulation results on fluid flow around rigid objects are shown. The intention is to demonstrate the flexibility with respect to structural shapes that can be used in the simulation. Coarse meshes have been used to show essential features of the enrichment and to demonstrate that the decoupling works for arbitrary fluid mesh densities.

For postprocessing purpose of scalar views, the intersected elements are shown using the triangular integration subdomains. They solely serve to properly display the sharp interface in the postprocessing tool. Standard elements would display a smeared solution, because the postprocessor has no information about the enrichment functions. Elements without a intersecting solid boundary are displayed as 4-node quadrilateral elements.

All results are postprocessed using Gmsh [42].

4.1 Flow against a Stationary Wall

The first example, shown in Figure 5 illustrates a flow around a structural corner. It can clearly be seen that there is no fluid flow within the structural domain in the lower...
right corner. The velocity along the fluid-structure interface is zero as prescribed using the Lagrange multipliers on the interface. The modified XFEM approach illustrated in Figure 3(b) was used for this simulation. It requires only 753 degrees of freedom (dof) compared to 1005 dof when all elements are calculated. This shows the importance of removing the interior fluid elements to save calculation time and memory.

4.2 Flow around a Thin Structure

In the second example, different flows fields develop around a thin structure, which thickness is smaller than the fluid element size. On the left side, a parabolic inflow is applied, while on the right some diagonal inflow enters the domain. It can be clearly seen in Figure 6 how the pressure and the velocities of the flow right and left of the structure are completely decoupled from each other and the small fictitious fluid domain between them.

The 3 sets of enrichments for the pressure are shown in Figure 7. The standard degrees of freedom, depicted in Figure 7(a) occupy most of the domain. Near the surface enrichment 1 and 2 overlap each other (Figure 7(b) and 7(c)). The summation of all enrichments gives the physical pressure field displayed in Figure 6.
Figure 6: Flow field around a stationary thin structure.

Figure 7: Enrichments used for the flow field around a stationary structure. The yellow dots indicate the enriched nodes used for the particular enrichment.
5 Conclusion

Fixed grid methods for fluid-structure interaction are subject of a growing number of current research undertaken. The ultimate goal is to remove the burden of fluid mesh movement and, if deformation of the structure becomes excessive, remeshing. For that purpose, we worked out a list of minimal requirements that have to be fulfilled before fixed grid methods can match or surpass ALE based methods.

In an attempt to meet these requirements, we proposed a new XFEM based approach, which builds on the achievements of already existing approaches, mainly the XFEM approach in [23] and the numerous works on the IB and DLM/FD methods. The main improvements compared to most IB and DLM/FD methods are the complete removal of the fictitious domain and a sharp interface description. Physical and fictitious domains are completely decoupled and most of the fluid unknowns from within the fictitious domain could be removed from the fluid system of equation. We did not require the additional level-set field used in [23]. The use of an partitioned instead of a monolithic approach allows us to study the fluid discretization in a simplified manner. Further studies will focus the FSI coupling, the extraction of the surface forces along the fluid-structure interface, the flow around sharp corners and a possible extension to higher order elements required for accurate solutions within the fluid domain.

REFERENCES


