The Nitsche's Method of the Navier-Stokes Equation for Immersed and Moving Boundaries

J. Benk*, M. Mehl*,** and M. Ulbrich*
benk@in.tum.de, mehl@in.tum.de, mulbrich@ma.tum.de

* Technical University of Munich, Germany.
** Institute of Advanced Studies (IAS), Germany.

Abstract: Imposing boundary condition on complex geometries for fluid dynamics requires special numerical methods. In the classical approach a conform unstructured mesh is created, that fits the complex geometry, which usually a no-slip condition is imposed on. This requires special meshing methods to create such a mesh. For massive parallel computations the meshing might pose a significant computational bottleneck. In this paper we present an other way of imposing boundary conditions for the Navier-Stokes equation on complex geometries. Instead of using a conform mesh we rely on parallel adaptive Cartesian mesh, which are not conform with respect to the geometry and are easy to decompose for parallel computations. This mesh is used in combination with the Nitsche's method. This approach implies a special treatments of the elements intersected by the geometry. We present in this paper the computed two- and three-dimensional benchmark results, which validates our approach.

Keywords: Numerical Algorithms, Computational Fluid Dynamics, Nitsche's Method, Immersed Boundary.

1 Introduction

Nitsche's method for the Poisson equation was introduced in [1] and is independent of the underlying mesh. For this reason this method is widely used in the free-mesh context. Since the Nitsche's method is partial differential equation (PDE) dependent it needs to be derived for each specific PDE. For the Navier-Stokes equation the Nitsche's method was presented in [2], but it was only applied in the classic way (on the facets of the elements). In this paper we present our approach to use the Nitsche's method in the combination with adaptive Cartesian mesh in two- and three-dimensional setting, where the boundary can intersect the mesh cells in a free and continuous way. In our previous work [4] we verified this approach by computing a two-dimensional stationary benchmark scenario. Here, we extend this approach to time variant and three-dimensional cases, where the boundary is even allowed to be moving.

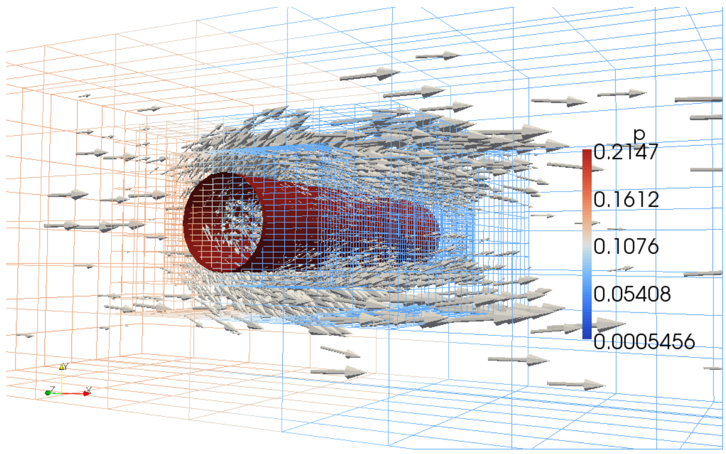

Figure 1: Three-dimensional benchmark results. Flow around a cylinder, which is represented by a triangulated surface.

2 Problem Statement

The dimensionless weak form is considered as the starting point of the Nitsche's method. For the Navier-Stokes equation this method consist of adding several domain and boundary integral terms (e.g. penalty) to this weak formulation, which enforce the Dirichlet boundary condition in a weak sens. For further details on the formulation we refer to [2, 4]. We present our methods to handle the cells which are cut by the boundary, so that we can accurately compute the integrals only on the computational domain Ω, and on the boundary Γ. Our approach is also well suited for parallel computations, since the underlying adaptive Cartesian mesh is rather easy to parallelize. All the presented methods are implemented in the Sundance FEM-based PDE-toolbox [3], which we also used for all the numerical computations. One example of the numerical computations is shown on Fig. 1, where the stationary three-dimensional benchmark was computed.

References

[1] J. Nitsche. Über ein Variationsprinzip zur Lösung von Dirichlet-Problemen bei Verwendung von Teilräumen, die keinen Randbedingungen unterworfen sind. *Abhandlungen aus dem Mathematischen Seminar der Universität Hamburg*, 0025-5858.

[2] R. Becker. Mesh adaptation for Dirichlet flow control via Nitsche's method, *Communications in Numerical Methods in Engineering.*

[3] K. Long, R. Kirby, and B. van Bloemen Waanders. Unified Embedded Parallel Finite Element Computations via Software-Based Fréchet Differentiation, *SIAM Journal on Scientific Computing.*

[4] J. Benk, M. Mehl and M. Ulbrich, Sundance PDE Solvers on Cartesian Fixed Grids in Complex and Variable Geometries, *Proceedinggs of the ECCOMAS Thematic Conference CFD & Optimization, Antlya, Turkey, May 23-25, 2011.*