Computers and Fluids 127 (2016) 182–193

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

A strategy to interface isogeometric analysis with Lagrangian finite elements—Application to incompressible flow problems

Raheel Rasool, Callum J. Corbett, Roger A. Sauer*

Aachen Institute for Advanced Study in Computational Engineering Science (AICES), RWTH Aachen University, Templergraben 55, Aachen 52056, Germany

ARTICLE INFO

Article history:
Received 10 April 2015
Revised 14 September 2015
Accepted 29 December 2015
Available online 7 January 2016

Keywords:
Incompressible Navier–Stokes
Isogeometric analysis
Enriched finite elements
SUPG/PSPG based stabilized FEM
Internal and external fluid flow

ABSTRACT

Motivated by the superior accuracy and better stability of isogeometrically enriched finite elements — when compared to standard Lagrangian finite elements for problems involving contact and debonding [15,16] — we extend their applicability to fluid flow problems. Internal and external flow involving incompressible Newtonian fluids is analyzed in the framework of the Finite Element Method (FEM). The concept of isogeometric analysis is applied only at certain localized regions while the bulk fluid is modeled with Lagrangian finite elements. This is achieved by using isogeometrically enriched finite elements that have a NURBS surface representation on one face while all other basis functions are represented by Lagrange polynomials. In this manner an enriched representation and analysis of the near surface region is possible, resulting in an approach that shows similar accuracy as the isogeometric analysis (IGA) while at the same time incurring similar cost as the standard FEM. This is demonstrated through several numerical examples involving laminar fluid flow.

1. Introduction

Solving the incompressible Navier–Stokes equations presents a challenging task for researchers analyzing problems involving fluid dynamics. Because of the inherent non-linearity due to the convective transport term, obtaining an exact solution for many physical flow problems is impossible. Hence over the years, several computational algorithms have been proposed to approximately solve these equations. One of the first solution algorithm for these equations can be credited to Harlow and Welsch [32] for their MAC grid method. This was followed by the so-called segregated algorithms which were based on pressure projection and solution of pressure Poisson equation [37,46]. Meanwhile, advanced algorithms which were motivated from compressible flow computations, such as the artificial compressibility method (ACM), were also applied to incompressible flow computations [10,45]. These methods, unlike the segregated algorithms, have the benefit of yielding a fully coupled and implicit system of equations for the momentum and mass balance laws.

Another area where computational modeling of Navier–Stokes equations has been actively pursued is in the finite element method (FEM) community. This was motivated by the success and wide-spread acceptance of FEM in solid/structural mechanics. However very early in its development it became apparent that the standard FEM, also known as the Galerkin method, is susceptible to numerical instabilities when applied to convection dominated problems. These instabilities can only be avoided with intense mesh refinement, which undermines the applicability of the method for practical usages.

Considerable research has been performed in developing stabilization schemes for FEM that would yield a stable and robust formulation. Early efforts in this regard comprised the so-called upwinding technique which amounts to adding artificial viscosity to the convective term. This treatment results in a stable but overly diffusive solution. Moreover, it leads to a formulation which is inherently inconsistent. A consistent approach to obtaining a stabilized finite element formulation for the Navier–Stokes equations was first presented in [9]. The method, known as the Streamline Upwind/Petrov Galerkin (SUPG), consists of adding an element level integral to the Galerkin formulation. Moreover, this integral is taken as a function of the residual of the momentum balance equation, thus resulting in a scheme which is consistent. The introduction of the SUPG formulation led to several developments in the context of consistently stabilized FEM and over the years several enhancements and variations have been proposed to this original idea (see [12,20,30,33,53]). An approach to systematically obtain various stabilized formulations was proposed in [34]. The method, known as the variational multiscale method (VMS), proceeds by separating the flow features into resolved and unresolved scales in a predetermined manner. Although initially proposed as a

* Corresponding author. Tel.: +492418099129.
E-mail address: sauer@aices.rwth-aachen.de (R.A. Sauer).

http://dx.doi.org/10.1016/j.compfluid.2015.12.016
0045-7930/

theoretical justification for the stabilized methods, VMS has since found wide acceptance for turbulent and complex flow analysis [2,4,5,27,39].

Apart from modifying the formulation itself, another strategy to improve a FEM scheme is to enrich the spaces from which the element basis functions are chosen. In [3] it was demonstrated that enriching the Galerkin basis functions with bubble functions, which are only defined in the element interior and vanish on the element boundaries, results in a stabilized formulation for convection-diffusion problems. In fracture mechanics, the extended finite element method (XFEM) [6] makes use of a strategy where the elements near the crack are locally enriched with functions that are able to capture a non-smooth field. The concept of isogeometric analysis (IGA), introduced in [35], uses NURBS and T-Spline CAD entities to define the element basis instead of classical Lagrange interpolatory polynomials. The strategy not only provides a geometrically exact model for computational analysis, but can also lead to a basis that may ensure higher continuity over the entire domain as compared to classical FEM.

Interest in improved geometrical representation for boundary surfaces is not a recent phenomena in FEM. Earlier attempts made use of the so-called curved finite elements where edges of the element were enriched with higher-order Lagrange polynomials to give better approximation of the underlying geometrical surface (see [22,28,58]). Recently, localized surface enriched elements based on higher-order Lagrange and Hermite polynomials were used in [47,48] for problems involving adhesion and multibody contact. The use of Hermite enriched elements renders a fully C^1-continuous surface, however their application is restricted to two-dimensional domains. The NURBS-enhanced finite element method (NEFEM) [49] offers another strategy where the boundary of the domain is mapped to a CAD surface description while the interior volume is modeled with standard piecewise finite elements. Unlike traditional FEM and IGA, the isoparametric concept in NEFEM is compromised since the NURBS basis is employed only for the geometrical representation while the solution field is approximated by standard Lagrange basis. This requires special numerical integration rules for the enriched elements.

Although higher continuity across the entire computational domain is an attractive property, it is however not always desirable to retain this feature. For fluid flows inherent with evolving interfaces, such as moving shocks or multiphase flows, a jump in the solution field is a physical reality. Having C^1- or higher continuity across such interfaces will lead to smearing of the jump in the solution field. In the context of IGA, this can be remedied by locating patch boundaries at such interfaces. This however is not trivial for the case of evolving interfaces, such as developing shocks, where the location of the shock wave is not known a priori. On the other hand, classical Lagrange finite elements coupled with interface modeling algorithms [25,51,54] present a much more convenient remedy. Additionally, higher level of continuity incurs an adverse effect on the performance of the linear solvers. In [13,14] it was observed that for the same degrees of freedom and basis order, a significant increment in computational cost is incurred when the continuity is raised. Such limitations motivate the development of a strategy where IGA can be employed in conjunction with classical Lagrangian finite elements.

In this paper, we present a novel strategy for the enrichment of a finite element based spatial description. We make use of the isogeometrically enriched finite elements which were first introduced in [15] as a multidimensional extension of the enrichment technique proposed in [47,48]. Similar to NEFEM, these elements are also enriched with a CAD based surface description, however unlike NEFEM the concept of isoparametric finite elements is retained by employing the same enriched basis for the geometry as well as the solution field. In this way, not only the requirement of special quadrature rules is circumvented, but the strategy ensures localized enrichment of the solution space as well. Moreover, since the enriched elements retain the characteristics of IGA, we demonstrate their applicability to act as an interface between classical Lagrangian finite elements and IGA elements. Such a connection yields a platform where IGA can be applied locally (e.g. near wall regions in flow problems where gradients are large) while the bulk domain is modeled with standard finite elements, effectively keeping the overall cost of the formulation at minimum. To the best of our knowledge such an avenue, where IGA can be blended with classical finite elements for flow problems, has not been explored previously.

The applicability of the proposed discretization strategy to fluid flow problems is explored in this study. As a natural first step laminar flow problems, having either an analytical solution or credible benchmarking data, are considered. A practical extension to turbulent flow modeling motivates the outlook of this research. The remainder of this paper is arranged as follows: In Section 2, the framework for the incompressible fluid flow solver is laid out. A brief overview of the surface enrichment methodology is given in Section 3. A strategy for imposition of non-homogeneous Dirichlet boundary conditions, which — unlike for interpolatory basis — is non-trivial for NURBS, is also discussed in this section. Numerical examples are given in Section 4 where the enrichment strategies are compared with standard FEM while Section 5 concludes this paper.

2. Framework for the flow model

The theoretical framework for a generalized unsteady incompressible fluid flow model is discussed in this section. It begins with the description of the governing equations, which describe the momentum and mass balance for the entire system. The equations are first expressed in strong form. Their variational (weak) description follows consequently. The section concludes with a description of the resulting finite element formulation which is discussed in conjunction with the stabilization technique used in this study.

2.1. Governing equations

At a given time $t \in [0, T]$, let us consider a spatial domain $\mathcal{B} \subset \mathbb{R}^d$, where d is the number of spatial dimensions. The boundary of the domain is given by $\partial\mathcal{B}$ such that $\partial\mathcal{B} = \partial_v\mathcal{B} \cup \partial_t\mathcal{B}$, where $\partial_v\mathcal{B}$ represents the Dirichlet boundary, and $\partial_t\mathcal{B}$ denotes the Neumann boundary. Then within $\mathcal{B}$, for an incompressible fluid, the momentum and mass conservation laws dictate that

$$\rho\, \boldsymbol{a} - \nabla \cdot \boldsymbol{\sigma} - \rho\, \boldsymbol{b} = \boldsymbol{0}, \tag{1}$$

$$\nabla \cdot \boldsymbol{v} = 0\,. \tag{2}$$

Here $\boldsymbol{b}$ represents the body forces acting on $\mathcal{B}$ while $\boldsymbol{\sigma}$ denotes the Cauchy stress tensor. For a Newtonian fluid, $\boldsymbol{\sigma}$ is given by the constitutive law

$$\boldsymbol{\sigma} = -p\,\mathbf{I} + 2\,\mu\,\nabla^s\boldsymbol{v}\,, \tag{3}$$

where μ is the dynamic viscosity of the fluid and $\nabla^s\boldsymbol{v} = \frac{1}{2}(\nabla\boldsymbol{v} + \nabla\boldsymbol{v}^{\mathrm{T}})$ is the symmetric part of the velocity gradient. $\mathbf{I}$ is an identity matrix with dimensions $(d \times d)$. Eqs. (1)–(3) constitute the incompressible Navier–Stokes equations. The acceleration $\boldsymbol{a}$ of a fluid particle in Eulerian framework is given as

$$\boldsymbol{a} = \frac{\mathrm{D}\boldsymbol{v}}{\mathrm{D}t} = \frac{\partial \boldsymbol{v}}{\partial t} + \boldsymbol{v} \cdot \nabla \boldsymbol{v}\,. \tag{4}$$

The model is closed by the following initial and boundary conditions for the velocity and the stress field:

$$\boldsymbol{v} = \boldsymbol{v}_o(\boldsymbol{x}) \quad \forall\, \boldsymbol{x} \in \mathcal{B} \text{ at } t = 0, \tag{5}$$