ARTICLE IN PRESS

Available online at www.sciencedirect.com

ScienceDirect

Mathematics and Computers in Simulation ▮ (▮▮▮▮) ▮▮▮–▮▮▮

MATHEMATICS AND COMPUTERS IN SIMULATION

www.elsevier.com/locate/matcom

Original articles

Isogeometric analysis for turbulent flow☆

Bohumír Bastl, Marek Brandner, Jiří Egermaier*, Kristýna Michálková, Eva Turnerová

European Centre of Excellence New Technologies for the Information Society, University of West Bohemia in Pilsen, Univerzitní 22, 306 14 Plzeň, Czech Republic

University of West Bohemia, Department of Mathematics, Univerzitní 22, 306 14 Plzeň, Czech Republic

Received 8 October 2014; received in revised form 7 December 2015; accepted 4 May 2016

Abstract

The article is devoted to the simulation of viscous incompressible turbulent fluid flow based on solving the Reynolds averaged Navier–Stokes (RANS) equations with different $k - \omega$ models. The isogeometrical approach is used for the discretization based on the Galerkin method. Primary goal of using isogeometric analysis is to be always geometrically exact, independent of the discretization, and to avoid a time-consuming generation of meshes of computational domains. For higher Reynolds numbers, we use stabilization SUPG technique in equations for k and ω. The solutions are compared with the standard benchmark example of turbulent flow over a backward facing step.

Keywords: Isogeometric analysis; Turbulent flow; $k - \omega$; Navier–Stokes equations

1. Introduction

The main goal of our work is to propose and implement the numerical model for solving turbulent flow based on the Galerkin approach with NURBS basis functions. We show that it is possible to effectively solve turbulent flow described by the Navier–Stokes equations with $k - \omega$ turbulent model by the isogeometric manner.

The objectives of isogeometric analysis based on NURBS (non-uniform rational B-splines) are to generalize and improve finite element analysis. It means to provide more accurate modelling of geometries and to exactly represent shapes such as circles, cylinders, ellipsoids, etc. Due to exact geometry at the coarsest level of discretization it is possible to eliminate geometrical errors. It also much simplifies mesh refinement of industrial geometries by eliminating communication with the CAD description of geometry. Further refinement of the mesh or increasing the order of basis functions is very simple, efficient and robust. At the same time, isogeometric analysis has many

☆ This work has been supported by Technology agency of the Czech Republic through the project TA03011157 "Innovative techniques for improving utility qualities of water turbines with the help of shape optimization based on modern methods of geometric modelling".

* Corresponding author at: European Centre of Excellence New Technologies for the Information Society, University of West Bohemia in Pilsen, Univerzitní 22, 306 14 Plzeň, Czech Republic.

E-mail addresses: bastl@kma.zcu.cz (B. Bastl), brandner@kma.zcu.cz (M. Brandner), jirieggy@kma.zcu.cz (J. Egermaier), kslaba@kma.zcu.cz (K. Michálková), turnerov@kma.zcu.cz (E. Turnerová).

http://dx.doi.org/10.1016/j.matcom.2016.05.010

0378-4754/

Please cite this article in press as: B. Bastl, et al., Isogeometric analysis for turbulent flow, Math. Comput. Simulation (2016), http://dx.doi.org/10.1016/j.matcom.2016.05.010

features in common with finite element analysis. For example the isoparametric concept in which dependent variables and the geometry share the same basis functions. Then the mesh, and the corresponding basis, can be refined and order-elevated while maintaining the original exact geometry. Then the isogeometric methodology can be a useful tool for computational fluid dynamics, in particular, turbulent flows.

High Reynolds number turbulent flows are important in many applications. Turbulent flows involve multiscale space and time-developing flow physics. The dynamics of all relevant scales of the flow described by the Navier–Stokes equations can be solved by the direct numerical simulation (DNS) approach, which is too expensive for most practical flows. Therefore the most common approach is the Reynolds-Averaged Navier–Stokes (RANS), which simulates the mean flow and effects of all turbulent scales. The efficient intermediate approach is large eddy simulation (LES), which can simulate significant flow unsteadiness that RANS cannot handle. In the absence of universal turbulence theory there exist many developments and improvements of the schemes including the empirical information.

2. Navier–Stokes equations

The model of viscous flow of an incompressible Newtonian fluid can be described by the Navier–Stokes equations in the common form

$$\begin{aligned} \frac{\partial \boldsymbol{u}}{\partial t} + \nabla p + \boldsymbol{u} \cdot \nabla \boldsymbol{u} - \nu \Delta \boldsymbol{u} &= \boldsymbol{f}, \quad \text{in } \Omega \times \langle 0, T \rangle, \\ \nabla \cdot \boldsymbol{u} &= 0, \quad \text{in } \Omega \times \langle 0, T \rangle, \end{aligned} \tag{1}$$

where $\Omega \subset \mathbb{R}^d$ (dimension $d = 1, 2, 3$) is the computational domain, $T > 0$ is the final time, $\boldsymbol{u} = \boldsymbol{u}(\boldsymbol{x}, t)$ is the vector function describing flow velocity, $p = p(\boldsymbol{x}, t)$ is the pressure function, ν describes kinematic viscosity and f additional body forces acting on the fluid. We do not assume only very small Reynolds numbers, but there are still some "limits" for which this model gives reasonable solution or it is necessary to use very fine discretization. The initial–boundary value problem is considered as the system (1) together with a suitable initial conditions and the following boundary conditions

$$\begin{aligned} \boldsymbol{u} &= \boldsymbol{w} \quad \text{on } \partial\Omega_D \times \langle 0, T \rangle \text{ (Dirichlet condition)}, \\ \nu \frac{\partial \boldsymbol{u}}{\partial \boldsymbol{n}} - \boldsymbol{n} p &= \boldsymbol{0} \quad \text{on } \partial\Omega_N \times \langle 0, T \rangle \text{ (Neumann condition)}. \end{aligned} \tag{2}$$

If the velocity is specified everywhere on the boundary, then the pressure solution is only unique up to a hydrostatic constant.

The Navier–Stokes equations describe turbulent incompressible flow. This flow contains many eddies of different sizes which are changing in time. The numerical methods for solving turbulent models are divided into the following categories:

- **Direct numerical simulation (DNS)** The average flux and all turbulent fluctuations are computed. It means, that we use FEM (or FVM etc.) method directly to solve the Navier–Stokes equations. It is necessary to use a very fine mesh to compute small turbulent fluctuations of the flow and a short time step for non-stationary problem. Therefore this approach is very computationally expensive.
- **Reynolds Averaged Navier–Stokes (RANS)** This approach simulates only average flux and effects of this flux to the flow. It uses the time averaged Navier–Stokes equations. The special term appears in the equations which is approximated by the appropriate approaches. The most common approaches are $k - \varepsilon$ or $k - \omega$ models. RANS is very often used in practice.
- **Large Eddy Simulation (LES)** LES simulates behaviour of large eddies which is realized by averaging of the Navier–Stokes equations in space dimension thus the small eddies are not considered. The behaviour of small eddies is described by the so called subgrid scale model, which can be for example computed by RANS.

3. RANS models

RANS is the most common model for solving the Navier–Stokes equations including turbulence. It is based on decomposition of the solution into the time-averaged value and fluctuation value. In two dimensions, the solutions

Please cite this article in press as: B. Bastl, et al., Isogeometric analysis for turbulent flow, Math. Comput. Simulation (2016), http://dx.doi.org/10.1016/j.matcom.2016.05.010