Flow Measurement and Instrumentation 60 (2018) 124–133

Contents lists available at ScienceDirect

Flow Measurement and Instrumentation

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

Hybrid simulation of a segmental orifice plate

M. Straka*, A. Fiebach, T. Eichler, C. Koglin

Physikalisch-Technische Bundesanstalt (PTB), Abbestr. 2-12, 10587 Berlin, Germany

A R T I C L E I N F O

Keywords:
CFD
SBES
LDA
Segmental orifice plate

A B S T R A C T

Computational fluid dynamics (CFD) has become a popular tool in the development of flow meters as an alternative to cost-intensive prototyping while preparing for approval tests or particular disturbances in pipe systems. In practical CFD applications, a turbulence model is required to predict the influence of turbulent flow features. Unfortunately, even the most state-of-the-art and commonly used turbulence models – the Reynolds-averaged Navier-Stokes (RANS) models – perform rather poorly in complex and separated flows, while scale-resolving methods such as large eddy simulations (LES) remain unfeasible for most industrial applications and higher Reynolds numbers. To make use of LES accuracy and current computing capacities nonetheless, the utilization of hybrid RANS-LES methods represents an efficient solution today.

In this paper, we present a modified stress-blended eddy simulation (SBES) turbulence model capable of making reliable predictions of disturbed pipe flows and the resulting effects on flow meter measurements demonstrated by simulating a segmental orifice plate and its influence on an ultrasonic meter. To illustrate the differences, the SBES results are compared to those of a standard k-ω RANS model. Typical mesh requirements and solver settings are discussed. Also, a spatially dependent blending function for SBES is introduced. Both simulations are compared to a large number of laser Doppler anemometry (LDA) and ultrasonic clamp-on measurements performed on a gravimetrically traceable flow test facility. The comparison with LDA shows the clear superiority of SBES, where RANS fails due to a massive overestimation of the recirculation zone. Near the orifice, SBES even has an advantage over LDA in predicting the ultrasonic meter performance.

1. Introduction

The inlet flow conditions have a large influence on the measuring accuracy of flow meters. If the flow profile deviates strongly from the fully developed calibration conditions – meaning a rotationally symmetrical and swirl-free velocity distribution – they have to measure the flow rate within a certain error range, despite the flow conditions at their point of use. In this regard, a number of tests with non-ideal (disturbed) flow conditions are specified in the standards for heat and cooling meters within the approval process (EN 1434–4 [7], ISO 4064–2 [13]). In industrial and domestic piping systems, disturbances are caused, for example, by bends, valves, reductions and other components. At flow test facilities, disturbed flows are usually emulated by means of standardized disturbance generators, because an installation with real bend configurations normally cannot be realized due to a lack of space or the usage of a clamping mechanism.

In preparation for approval tests with disturbance generators or particular flow disturbances in pipe systems, the development of flow meters is usually accompanied by time-consuming and cost-intensive prototyping. However, as a result of constantly increasing computing capacities, numerical methods such as computational fluid dynamics (CFD) present an increasingly attractive alternative with a considerable reduction of costs and time. Compared to measurements on a test rig, the CFD simulation cannot only predict the resulting flow rate deviations, but provides a profound understanding of the relevant phenomena inside the meter by visualizing the flow field. This allows an optimization of the internal geometry (possibly with flow-conditioning measures), or the derivation of suitable correction algorithms for certain flow conditions.

Unfortunately, the extent of turbulence modelling necessary for CFD simulations in industrial applications leads to a simplified and often distorted representation of complex flow phenomena. This applies in particular to the Reynolds-averaged Navier-Stokes (RANS) approaches, which still correspond to the industrial standard. While RANS provides engineering accuracy at low computational costs for a wide range of simple flows (Fröhlich and von Terzi [9]), it is known to have limitations in predicting a series of complex flow situations. This includes flows with strong swirl components, free shear layers or massive

* Corresponding author.
E-mail address: martin.straka@ptb.de (M. Straka).
URL: http://www.ptb.de/cms/ptb/fachabteilungen/abt7/fb-75.html (M. Straka).

https://doi.org/10.1016/j.flowmeasinst.2018.02.006
Received 18 August 2017; Received in revised form 16 January 2018; Accepted 11 February 2018
Available online 14 February 2018

separation.

Due to the calibration in accordance with the law of the wall, RANS models are basically well suited for less complex, wall-bounded pipe flows in the field of flow measurement applications. If, however, separation has a dominating influence, the agreement between the simulation and experiments is often less sufficient. For the RANS simulations of an orifice covering 7% of the pipe cross section, Weissenbrunner and Eichler [29] have found, for example, deviations on the order of up to 10% compared to laser-optical measurements. Also, Rockenstein [22] has demonstrated the inability of various RANS models to predict the strongly separated flows behind partially opened ball valves. Beyond the field of pipe applications, numerous studies have been performed to demonstrate RANS insufficiencies regarding benchmark test cases with separated flows, such as inclined flat plates (Breuer et al. [3]), turbine plates (Ranjan et al. [21]) or backward-facing steps with different step angles (Choi et al. [4]). These results are hardly surprising, considering that RANS provides information only for the mean quantities of the flow, while the entire turbulent spectrum is modelled, which in turn always includes a number of empirical constants.

In contrast to RANS, scale-resolving methods such as the large eddy simulation (LES) are able to resolve turbulent fluctuations up to a certain length and time scale. For this reason, LES is capable of making highly precise predictions for all kinds of flow problems. However, wall-bounded flows such as the flow in a pipe require very small time steps and extremely fine grid resolutions. In addition, the computational effort increases exponentially with an increasing Reynolds number (Baggett et al. [2]) as the boundary layer becomes increasingly thinner and, thus, increasingly smaller turbulence scales outside the viscous sublayer must be resolved. For the range of Reynolds numbers interesting under technical aspects, scale-resolving methods such as LES will, therefore, remain too cost-intensive and time-consuming for many years and are currently only playing a subordinate role in the development of flow meters and other industrial applications.

To use the precision of scale-resolving simulations for wall-bounded flows at high Reynolds numbers, a great number of hybrid RANS-LES methods - beginning with the detached eddy simulation of Spalart et al. [27] - have been developed in the past 20 years. Hybrids are based on the formal agreement of the RANS and LES equations which result from the RANS averaging and low-pass filtering of the Navier-Stokes equations. Depending on the model, the computational domain is divided into RANS and LES areas either based on the grid resolution, the proximity to the wall, or within defined zones. While areas with, e.g., free shear flows or swirl are covered by the LES model, boundary layers and parts of the domain, where RANS models are known to be reliable in predicting the relevant flow phenomena, e.g., at a reasonable distance from separation zones, can be calculated with comparatively coarse grids in RANS mode. This leads to a considerable reduction in computational time by at least a factor of 10 or higher, growing exponentially depending on the Reynolds number [17]. Satisfactory simulation results have been obtained for a variety of academic test cases and also industrially relevant applications, e.g., [12,10,20]. The wide spectrum of hybrids existing today results from the fact that each method is designed for a certain category of flow. An overview is given, e.g., by Fröhlich and von Terzi [9]. Hybrids with RANS boundary layers are generally referred to as wall-modelled large eddy simulations (WMLES) and are the preferred choice for hybrid pipe simulations. In contrast to LES, WMLES is quasi-Reynolds-independent with regard to the grid size (Menter [17]), as the turbulent scales in the proximity of the wall are represented by the modelled Reynolds stresses.

In this paper, we present a hybrid simulation method capable of predicting disturbed pipe flow applications and their effect on flow meters with LES quality at reasonable computational costs. For this purpose, a stress-blended eddy simulation model (SBES), which is generally capable of emulating a wide range of hybrid turbulence models and is implemented in commercial CFD codes like ANSYS Fluent [1], was set to operate in WMLES mode with a 2-equation RANS model in the near-wall region. The transition between the RANS and the LES turbulence model was specified by a spatially dependent blending function shielding both domains regardless of the mesh resolution.

Fig. 1. The segmental orifice plate. The lower third of the pipe's cross section is covered by a horizontal restriction.

The modified SBES model is demonstrated by performing a simulation of a segmental orifice plate (see Fig. 1) at a Reynolds number of 50,000 and by predicting the effects on ultrasonic flow meter measurements. The orifice is used as a disturbance generator to emulate velocity profiles similar to those downstream of a single 90° bend and therefore represents one of the most common disturbances in domestic and industrial piping systems. In spite of its rather simple geometry, the flow features of the orifice are well comparable with the above-mentioned cases in which no satisfactory results have been achieved with RANS, as the simulation must predict both the size of the recirculation zone as well as secondary velocity components correctly.

The paper is structured as follows. The numerical setup for the SBES model is dealt with in Section 2. Afterwards, the time-averaged results of the flow field behind the orifice are examined in Section 3. To demonstrate the differences, the SBES results are compared to a steady-state RANS simulation that was carried out using a standard k-ω model. Section 4 deals with the validation of the simulation results. Following a description of the experimental setup and a short introduction to a set of quantitative performance indicators, the numerical results are compared to laser Doppler anemometry (LDA) and ultrasonic clamp-on measurements performed on a gravimetrically traceable flow test facility.

2. The stress-blended eddy simulation turbulence model

2.1. Governing equations and model concept

CFD is a numerical method used to solve discretized differential equations describing the motion of fluid flows (for a detailed explanation of the common CFD terms used here, see, e.g., Ferziger and Peric [8]). In a transient simulation with a hybrid turbulence model, this includes a solution of the RANS and LES equations, which can be derived from the Navier-Stokes equations. For incompressible fluids, they consist of one continuity and three momentum equations. If the Boussinesq hypothesis is employed, the governing equations for LES and RANS are formally identical, which is the basis for hybrid turbulence modelling. Using Einstein notation, the continuity and momentum equations are given by

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \quad (1)$$