Chemical Engineering Journal 250 (2014) 175–189

Contents lists available at ScienceDirect

Chemical Engineering Journal

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

Effect of inlet conditions on the accuracy of large eddy simulations of a turbulent rectangular wake

Katrine M. Nilsen [a], Bo Kong [a], Rodney O. Fox [b], James C. Hill [b], Michael G. Olsen [a,*]

[a] Department of Mechanical Engineering, Iowa State University, Ames, IA, USA
[b] Department of Chemical and Biological Engineering, Iowa State University, Ames, IA, USA

HIGHLIGHTS

- A large eddy simulation is performed on a turbulent confined wake flow.
- Presimulations were performed to optimize inlet conditions and subgrid model.
- The simulation is validated against particle image velocimetry data.
- Detailed comparisons of one-point statistics as well as spatial correlations were made.
- With carefully selected inlet conditions, the simulation showed very good agreement with the experiment.

ARTICLE INFO

Article history:
Received 28 November 2013
Received in revised form 15 February 2014
Accepted 20 March 2014
Available online 28 March 2014

Keywords:
Turbulence
Large eddy simulation
Particle image velocimetry
Turbulent wake
Fluid mechanics

ABSTRACT

A large eddy simulation (LES) study was performed on a turbulent incompressible wake flow in a rectangular channel. The simulation results were evaluated using particle image velocimetry (PIV) data from a previous experimental study of the same flow (Liu et al., 2013). Comparisons were made of one-point statistics as well as spatial correlations.

An extensive pre-simulation study was carried out in which the effect of inlet conditions, grid resolution, time step and subgrid model was investigated and the parameters were optimized. Using the digital filter method of Klein et al. (2003), turbulent inflow velocities were generated based on velocity mean and variance obtained from the experimental data and correlation length scales. It was found that the simulation results in large parts of the domain were strongly dependent on the inlet length scales specified. With a suitable set of length scales, the inlet method was successful at providing inlet conditions that generated accurate simulation results.

The very good agreement seen between experiment and simulation demonstrates LES as a method that, with carefully selected inlet conditions, not only can predict the pointwise turbulence statistics of a liquid wake flow, but also capture key features of its large-scale turbulent structures.

1. Introduction

The wake is together with the jet and the mixing layer a commonly studied free shear flow. A wake is generated when a fluid flows over a body leaving a region of velocity deficit behind the body. The study of the turbulent wake is not only important for gaining understanding of turbulence, but also relevant in many engineering applications, such as aviation industry, chemical process industry and wind power engineering. In this context, and in the study of turbulent flows in general, computational fluid dynamics (CFD) has for many years been an important tool [35,40].

Common wake configurations are wake behind a bluff body [18,4,31] and wake behind a flat plate [11]. An incompressible confined rectangular wake has been studied experimentally by Liu et al. [22] and Feng et al. [5] who employed particle image velocimetry (PIV) and planar laser-induced fluorescence (PLIF) to measure instantaneous velocity and concentration fields in the wake, and studied in particular the mixing of a passive scalar. Liu et al. [22] simulated the confined rectangular wake flow using Reynolds-averaged Navier Stokes (RANS) method and compared the result to the experimental data with good agreement. Although RANS is a cost-effective numerical method, it has limitations in that instantaneous structures of the flow cannot be captured. Large eddy simulation (LES) is a computational method that aims to solve the transient Navier–Stokes equations for the larger energy

* Corresponding author. Tel.: +1 515 294 0073; fax: +1 515 294 3261.
E-mail address: mgolsen@iastate.edu (M.G. Olsen).

http://dx.doi.org/10.1016/j.cej.2014.03.069
1385-8947/

containing scales of a flow directly while modeling the smaller dissipative scales. Thus LES can provide very detailed flow information for turbulent scales down to the smallest resolved scale. LES is therefore the preferred alternative in many situations. In this work the confined rectangular wake is simulated using LES and PIV data are used for detailed assessment.

PIV is a non-intrusive method to measure instantaneous velocity fields [1]. The fluid is seeded with light scattering particles with small enough Stokes number that they accurately follow the flow. The flow is illuminated by a pulsating laser sheet, and a camera is used to capture images of the particles for each laser pulse. A correlation technique is used to determine the displacement of the particles for each picture pair. Thus PIV provides instantaneous two-dimensional velocity fields that are excellent for validation of CFD methods and offers the possibility of comparisons on a high level including spatial structure information.

In terms of complexity, the flow investigated lies between the simple flow configurations that are commonly used for testing and validation of CFD and the highly complex flow cases found in industry where only limited experimental data are available. Thus a successful utilization of CFD methods for this flow contributes to wider applicability and higher reliability of the methods.

A challenge in LES is setting the inlet condition. Large eddy simulation requires a turbulent inlet condition that ensures the right distribution of kinetic energy as well as coherence of the fluctuations [36]. For dissipative flows where little turbulence is generated inside the domain the result will heavily depend on the inlet condition specified. The traditional way of generating turbulent inflow velocities is to superimpose random fluctuations on a specified mean velocity. This method is easy to implement, but the drawback is that it produces uncorrelated inflow velocities and an improper energy spectrum. This causes the inflow turbulence to decay rapidly when entering the solution domain [42,23]. When the geometry allows for it, the inlet values can be obtained by applying a periodic boundary condition or by performing a precursor simulation of the region upstream of the inlet. To limit the extra computational cost of a precursor simulation, Li et al. [20] used a finite time series of the precursor simulation and recycled through the values to get continuous inflow. In cases where sufficiently detailed experimental values are available, the inlet condition can be based on these, as in the digital filter method of Klein et al. [16] where turbulent inlet conditions with specified first and second order statistics are generated. Unless the experimental data are of such resolution both in time and space that they can be used directly as inflow condition in LES, inflow generation necessarily includes modeling and assumptions. It is therefore highly important to evaluate the turbulent inlet condition employed. This should be done not only by looking at the generated inflow velocities themselves but also by assessing how they perform in a simulation by evaluating the simulation results that are obtained using the inflow velocities. In this work the geometry does not allow for periodic boundary conditions or precursor simulation. Instead, inlet velocities were generated using the digital filter method of Klein et al. [16] and PIV data from Liu et al. [22] and the performance of the method in a simulation as well as the sensitivity to the input parameters were investigated.

Another challenge in LES lies in modeling the effect of the unresolved scales. Many subgrid models have been suggested, defined in spectral space, see e.g. [3], or physical space. Examples of the latter are the Smagorinsky model [39], the one-equation model [28,44,13] and the structure function model [26]. Further information regarding subgrid models can be found in e.g. [19,8,36]. Even for simple flows, the performance of the subgrid models can vary greatly depending on the flow and modeling parameters. For example, Fureby et al. [8] compared different subgrid models in simulation of forced and decaying homogeneous isotropic turbulence and found that the difference in performance of the different subgrid models was dependent on the Reynolds number of the flow and the grid resolution.

Considering the challenges in LES it becomes clear that some form of validation of the entire simulation method is required. When evaluation of LES is based on direct numerical simulation (DNS) values, the same inlet conditions can be used in the LES simulations. In addition, DNS offers the advantage of fully resolved data that can be taken as the exact solution. A comparison with DNS values is therefore quite straightforward. However, DNS data are not readily available for complex flows. Validation using DNS data is limited to demonstrating how well LES performs on flows where DNS is already feasible [43]. Thus this approach has limited use.

Evaluation of LES results by comparison to experimental data widens the range of applicable flows since experiments can be performed on flows too complex for DNS. When comparing LES results to experimental values there are several aspects that must be considered. First, experimentally measured values are averaged values over a probe volume and thus are effectively filtered values [33]. LES gives the filtered field, and since an experiment essentially also gives filtered results, one might conclude that when the two filter sizes match, the comparison can be carried out directly. However, useful definitions for filter size are lacking both for LES and for experiments [15]. Moreover, comparison with experimental values requires a simulation inlet condition that corresponds to the inlet of the experimental flow. It is then clear that the assessment of LES is closely related to evaluating the inlet condition.

In the context of evaluation by comparison the question of what values to compare arises. Intuitive first choices are mean and variance. However, since LES provides detailed full field transient flow data, the comparisons can also include comparisons of temporal and spatial correlations, higher order statistical moments, and distribution functions [2,15,17].

In this work OpenFOAM (Open Field Operation and Manipulation) [32], an open source software package for CFD, was used to perform the simulations. OpenFOAM offers a wide range of CFD tools and is rapidly growing in use. A successful demonstration of this software package is therefore of great interest to many, both in research and in industry.

In the following section the flow geometry will be presented followed by a detailed description of the numerical method used to simulate the flow in the experiment. The results and comparisons are presented and discussed in Section 4. Finally, a summary is given and conclusions are drawn in Section 5.

2. Experimental setup

Fig. 1 shows the test section used in the experiments. The test section is a rectangular channel having a cross-section measuring 6 cm by 10 cm and a length of 1 m. The channel has three inlets separated by two splitter plates so that the width of each inlet is 2 cm. This distance will in the following be noted d and used as a characteristic length for normalizing purposes. Before the fluid enters the test section it goes through flow conditioning sections consisting of a packed bed and turbulence reducing screens followed by a 16:1 contraction. The flow rate in each stream was set to 1 L/s, corresponding to a bulk velocity, U_0, of 0.5 m/s. This gives a Reynolds number based on hydraulic diameter of the channel of 37,500.

The optical setup consists of a dual-pulse New Wave Gemini Nd:YAG laser with laser light wavelength of 532 nm and a 12-bit LaVision Flowmaster 3S CCD camera with a resolution of 1280×1024 pixels. The time delay between two laser pulses were set to 700 μs and the duration of each laser pulse was about 5 ns.