Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 127 (2015) 87 – 94

Procedia Engineering

www.elsevier.com/locate/procedia

International Conference on Computational Heat and Mass Transfer-2015

Detached Eddy Simulation of Turbulent Flow in Stirred Tank Reactor

Thiyam Tamphasana Devi[a], Bimlesh Kumar[b]*, Ajey Kumar Patel[c]

[a]National Institute of Technology Manipur, Imphal-795001, India
[b] Indian Institute of Technology Guwahati, Guwahati-781039, India
[c] National Institute of Technology Warangal, warangal-506004, India

Abstract

The difficulties associated with the use of the standard LES (Large Eddy Simulation) models, particularly in near-wall regions, has lead to the development of hybrid models that attempt to combine the best aspects of RANS (Reynolds Averaged Navier Stokes) and LES methodologies in a single solution strategy. An example of a hybrid technique is the detached-eddy simulation (DES). In this paper, application of DES model is made in the analysis of turbulent flow generated in impeller driven (Rushton Impeller) stirred tank reactor in order to understand the flow behavior especially around the impeller region and near the wall of the tank. The simulated result is compared with Experimental results of Wu and Patterson [1]. DES is found more accurate against the experimental result in predicting the turbulent flow of velocity profile than the LES model which ultimately increases its interest in turbulent flow analysis.

Peer-review under responsibility of the organizing committee of ICCHMT – 2015

Keywords: computational fluid dynamics; detached eddy; turbulent flow; stirred tank

1. Introduction

In the past recent years, the application of computational techniques like computational fluid dynamics (CFD) tool have been increasingly and widely used in the areas of analysis of fluid flows in stirred tank reactors. These techniques allows the user to predict the magnitude of fluid flows as well as its pattern of flow enhancing the user to

* Corresponding author. Tel.: +91-361-258-2420; fax: +91-0361-258-2440.
E-mail address: bimk@iitg.ernet.in

Peer-review under responsibility of the organizing committee of ICCHMT – 2015
doi:10.1016/j.proeng.2015.11.430

understand the possible characteristics of fluid flow pattern prior to the execution of the costly and laborious experimental work. These understanding of fluid flow pattern in stirred tank reactors are important for design, scale-up and optimization purposes of such stirred tanks. When it comes to in terms of prediction or simulation (as a substitute to the experimental work) of the fluid flow characteristics, the simulation or numerical parameters are the major concern to the accuracy of the predicted results as these techniques are purely based on numerical parameters. Prediction of turbulent flow in complex geometries of stirred tank where the moving boundaries exists like impeller region (near the impeller tip), at baffles and at walls create more trouble in accurate prediction and eventually concerns over the ideal selection of numerical parameters. Fluid near the impeller moving regions are the highest in magnitude as the fluid moves with the maximum rotational speed of impeller in such regions, and are highly turbulent and anisotropic which made the simulation more challenging. One of the most important numerical parameter to be carefully considered in the analysis of turbulent flow in stirred tank reactors is turbulence model. Among the available turbulence model, it is well-known from the literature that large-eddy simulation (LES) is able to better predict the time-averaged flow quantities, including those related to turbulence [2]. In a LES, a low-pass filtered version of the Navier-Stokes equation is solved. The fluid motion at the subfilter scales is taken care of by a model. It is a 3-D, transient numerical simulation of turbulent flow, in which the large flow structures are resolved explicitly and the effects of subgrid (or subfilter) scales are modeled, the rationale being that the latter are more universal and isotropic in nature. LES model of stirred-tank flow are computationally expensive. The computational cost of an LES is largely dictated by spatial resolution requirements. Away from walls, the spatial resolution needs to be such that the cut-off spatial frequency of the low-pass filter falls within the inertial subrange of turbulence. Issues with LES related to boundary layers led to the idea of formulating a turbulence model that is cheaper to run and better predict turbulent flows, called detached eddy simulation (DES) or hybrid (RANS-LES) turbulence model. The main idea of this approach is to perform LES away from walls where demands on resolution are not that strong, and revert to RANS modeling where LES is not affordable, i.e., in boundary layers. In strong turbulence, flow structures close to the wall are very small and anisotropic [3]. Thus, an LES would need a very fine grid within the boundary layer, which implies that the computational cost does not differ appreciably from that of a direct numerical simulation (DNS) [4]. In-adequate grid resolution of boundary layers can severely degrade the quality of a large eddy simulation. Therefore, DES was proposed by Spalart et al. [5] in an attempt to reduce the computational cost as well as to provide a good prediction of turbulent flows, containing boundary layers. A DES is an LES that transfers to a RANS-based simulation in boundary layers, thus permitting a relatively coarse grid near walls. To the authors' knowledge, there are only a very few studies of DES model in turbulent flow modeling in stirred tank exists [2,6,7].

The main objective of this work is to assess the quality of DES predictions for stirred-tank flow. For this, a detailed comparison with experimental data in the vicinity of the impeller was performed because in this region the flow is being generated, and here the effect of inadequate wall layer resolution would be most visible (as the flow is highly swirling and anisotropic in such region). Close to the impeller, boundary layers detaching from impeller blades and associated vortex structures dominate the flow. In addition to comparing the DES results with experimental data of Wu and Patterson [1], it also compares with LES model to judge the performance of DES model in the prediction of turbulent flow with respect to LES model.

Nomenclature

C	impeller clearance depth
d	diameter of impeller
H	height of water
N	impeller speed
r, z	radial and axial distance
T	diameter of tank
U_r	radial velocity
U_{tip}	impeller tip velocity