International Journal of Heat and Fluid Flow 67 (2017) 27–42

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Evaluation of numerical wall functions on the axisymmetric impinging jet using OpenFOAM

J.-A. Bäckar[a,*], L. Davidson[b]

[a] Department of Energy Conversion and Physics, Volvo Group Truck Technology, Gothenburg SE-405 08, Sweden
[b] Division of Fluid Dynamics, Department of Applied Mechanics, Chalmers University of Technology, Gothenburg SE-412 96, Sweden

ARTICLE INFO

Article history:
Received 16 December 2016
Revised 3 July 2017
Accepted 4 July 2017

Keywords:
Computational fluid dynamics
OpenFOAM
Impinging jet
Near-wall
Numerical wall function
Speed-up

ABSTRACT

Two new robust numerical wall functions are evaluated and the effect of different approximations used in earlier numerical wall functions by Craft et al. (2004) and by Bond and Blottner (2011) are demonstrated. A standard low-Reynolds-number turbulence (LRN) model is used as reference but with different meshing strategies. The objective is to considerably reduce the total central processing unit (CPU) cost of the numerical simulations of wall bounded flows while maintaining the accuracy of any LRN model.

When calculating turbulent flow problems, a tremendous speed-up may be achieved by decoupling the solution of the boundary layer from the bulk region by using a *wall function*. However, most wall functions are quite limited and based on assumptions which are not valid in complex, non-equilibrium flows.

The present wall functions solve full momentum and energy equations on a sub-grid, using face fluxes of advection and diffusion to transfer the solution to and from the sub-grid. The evaluation was carried out on an axisymmetric impinging jet using the turbulence model of Launder and Sharma (1974) with the correction of Yap (1987).Compared to standard LRN calculations, the results show perfect agreement to less than one-sixth of the computational cost. However, the reason for the speed-up is shown to come mainly from the meshing strategy, and none of the evaluated wall functions add much additional value.

1. Introduction

The use of CFD continues to increase in industry, due to the savings that can be achieved in both time and cost over corresponding experiments. To predict industrial flow problems, which often have a turbulent nature, the most common approach is still a Reynolds Averaged Navier Stokes (RANS) simulation together with a turbulence model. Considering accuracy and computational cost for a certain class of flows, dominated by boundary layer effects, the most important aspect of such simulations is how the boundary layer is treated.

The boundary layer is the fluid layer in the immediate vicinity of a wall, in other words, where the viscous effect is not negligible. It extends to the fully turbulent regime and, even though it only occupies a smaller part of the flow, this region may account for the majority of the computing time. The reason for this relatively high computational cost is that boundary layer flow properties change at a rate typically two or more orders of magnitude faster than elsewhere in the flow.

These high gradients require a very fine computational mesh in order to be resolved accurately. The family of turbulence models that uses this strategy of resolving the boundary layer is called low-Reynolds-number (LRN) models. These models use the same set of equations for all parts of the flow and may be accurate for most types of flows, but the resulting equation system converges slowly, especially at high Reynolds number. The turbulence models span from simple mixing-length schemes, through two-equation eddy-viscosity models of different complexity, to second-moment closure models.

To mitigate the slow convergence of the LRN models, the boundary layer and the fully turbulent region may be decoupled, thus acknowledging the different computational requirements for the two regions. The most common approach is the high-Reynolds-number (HRN) model together with a "wall function", which uses a coarse mesh where the first cell layer covers the inner boundary layer, including the inner part of the log-layer. Instead of solving partial differential equations on a fine mesh, an analytical expression is used to model the flow in the boundary layer. HRN mod-

* Corresponding author.
E-mail address: jon-anders.backar@chalmers.se (J.-A. Bäckar).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2017.07.004
0142-727X/

els with wall functions are often less accurate, and they are also sensitive to the mesh resolution close to the wall. Attempts have also been made to analytically integrate the transport equations, but these give restrictions on the geometry to allow for analytical integration.

A more advanced way of decoupling the boundary layer from the fully turbulent region is to use a "numerical wall function". This wall treatment can be seen as a hybrid of HRN and LRN modelling where the boundary layer is indeed resolved but with a slightly simplified set of partial differential equations compared to what is used in the rest of the domain.

The first numerical wall function in a RANS context, called UMIST-N, was developed by Gant (2002) and Craft et al. (2004). They divided the wall-adjacent cells into a sub-grid where simplified RANS equations, using some sort of boundary layer assumptions, were solved, including tangential velocity and turbulent quantities. Furthermore, the pressure gradient was assumed to be constant in the wall-normal direction over the sub-grid and could hence be interpolated from the main-grid. A Dirichlet condition, with interpolated values from the main-grid's first and second wall-adjacent cells, was set on the boundary of the sub-grid, opposite to the wall, for all solved quantities. The calculated wall shear stress, averaged turbulent production and dissipation terms from the sub-grid were then used to replace the corresponding terms in the main-grid equations. This yielded results close to a default LRN solution at computing times of an order less in magnitude.

A few studies have investigated variations of the UMIST-N model. Myers and Walters (2005) simplified the sub-grid equations even further by using a linear profile for the wall-normal velocity and used the 2-D continuity equation to calculate the stream-wise velocity gradient. The convection was neglected in the turbulence equations. In this way, the 2-D boundary layer equations were reduced to 1-D equations for the tangential velocity and turbulent model quantities. Bond and Blottner (2011) proposed a similar model for compressible and transient flow by neglecting convection in all transport equations. Chedevergne (2010) also developed a similar 1-D model but implemented it in an unstructured code where the sub-grid only covered the main-grid's wall-adjacent cells from the wall up to the centroid of the main cells. He also included compressibility terms in the model equations. Lastly, Wald (2016) tried to adapt the UMIST-N for a second-moment closure turbulence model which gave similar results in accuracy as Craft et al. reported earlier on an axisymmetric impinging jet. However, Wald (2016) also concludes that the model is unstable and chooses not to pursue with other geometries. It is not clear from his thesis whether the robustness issues arise from the use of UMIST-N itself or only in combination with the turbulence model used.

Even though the processing speed of computers is continuously increasing, the CFD community is generally far from satisfied with available computing resources, regardless of whether they act in industry or elsewhere. As e.g. Spalart (2000) describes, HRN and LRN modelling belong to the simpler variants of methods that solve turbulent transport equations. Nevertheless, with the use of these relatively simple models for large and complex problems, the computational resources often set a limit to what can be done. If the same models are used in design-of-experiments or optimisation loops, the computing resources will obviously always be a limitation to what can be achieved with simulations for the next decades.

With this background, it is important to acknowledge and deploy turbulence modelling techniques that offer the best compromise between accuracy and computing requirements. The numerical wall function strategy deployed in RANS modelling has existed since at least 2004 but has not yet been widely adopted by the CFD community despite its excellent features of supplying a sweet-spot between HRN and LRN modelling. The most important reasons for this are probably the cost of implementation and the close connection to the turbulence model. To support a turbulence model, earlier numerical wall functions need to implement each model's specific source and sink terms, making the implementation and maintenance more awkward.

The purpose of this investigation is twofold: first to make an implementation in an openly available and unstructured CFD code and relax the dependence between the implementation and specific supported turbulence models.[1] The second purpose is to evaluate different near-wall strategies including commonly used assumptions in earlier numerical wall functions.

Two new numerical wall functions are built upon the work from Craft et al., but they use a more mathematically stringent coupling which is independent on choice of turbulence model. This has been achieved by an innovative use of face fluxes, making a two-way connection between the main-grid and the sub-grid. These new numerical wall functions are evaluated on a turbulent axisymmetric impinging jet with and without assumptions made in earlier numerical wall functions, but also with standard integration to the wall using a similar mesh cell distribution which is normally used in numerical wall functions. It is found that an advanced mesh strategy gives a similar speed-up as a decoupled approach, i.e. the numerical wall function, and we demonstrate that it is also the most robust alternative. Thus, in this study, no added value is found for the concept of the numerical wall function.

2. Method

The effect of different meshing strategies is first investigated using a standard wall treatment, i.e. integration to the wall, used with LRN turbulence models. Second, the implementation of the robust wall functions is verified. Third, the new wall functions are evaluated regarding their sensitivity with respect to how far the interface is placed from the wall in y^+ units. Last, the effect of the assumptions in earlier numerical wall functions Craft et al. (2004) and Bond and Blottner (2011) are compared with the new robust wall functions and to a standard LRN set-up deploying the same local mesh density as with the numerical wall functions.

All tests and implementations have been done in OpenFOAM® (2015), Open Field Operation and Manipulation, CFD Toolbox, which is a free and open source CFD software package. It uses a co-located methodology on unstructured polyhedral meshes. This methodology is used in both the main-grid and the sub-grid of the numerical wall functions. However, a restriction is introduced for the wall-adjacent cells in the main-grid to be prismatic.

2.1. Governing equations

The full 3D-RANS equations are solved in both the main-grid and the sub-grid, with the assumption that the pressure gradient in the wall-normal direction is constant in the sub-grid. Incompressible Reynolds averaged Navier–Stokes in tensor notation reads

$$\bar{u}_{i,i} = 0, \tag{1}$$

$$\bar{u}_{i,t} + (\bar{u}_i\bar{u}_j)_{,j} = -\frac{\bar{p}_{,i}}{\rho} + \left[(\nu+\nu_t)(\bar{u}_{i,j}+\bar{u}_{j,i})\right]_{,j} \tag{2}$$

$$T_{,t} + (\bar{u}_i T)_{,i} = \left[\left(\frac{\nu}{\sigma}+\frac{\nu_t}{\sigma_t}\right)T_{,i}\right]_{,i} \tag{3}$$

with the LRN model of Launder and Sharma (1974) including the correction of Yap (1987),

$$k_{,t} + (\bar{u}_i k)_{,i} = \left[\left(\nu+\frac{\nu_t}{\sigma_k}\right)k_{,i}\right]_{,i} + P_k - \tilde{\varepsilon} - D \tag{4}$$

[1] A cleaned up version of the implementation is planned to be published under https://github.com/backar.