Wall-modeled LES of turbulent flow over a two-dimensional Gaussian bump

Prahladh S. Iyer* and Mujeeb R. Malik**
Corresponding author: prahladh.iyer@nianet.org

* National Institute of Aerospace, USA.
** NASA Langley Research Center, USA.

18 January, 2022

Abstract: We perform wall-modeled large eddy simulations (WMLES) of turbulent flow over a nominally two-dimensional Gaussian-shaped bump geometry to assess its performance in the accelerating and separation regions of the flow. The flow conditions are based on high-fidelity numerical simulation of Uzun & Malik (*AIAA Journal 2021*) and the CFD validation experiments of Williams et al. [*AIAA SciTech Paper 2020-0092*]. The oncoming flow Mach number is 0.2, with the bump length-based Reynolds number (Re_L) of 2 million. In our previous study, while WMLES with the constant coefficient Vreman subgrid scale model and equilibrium wall model performed satisfactorily at lower $Re_L = 1$ million, it failed to predict flow separation at the higher Re, contrary to the experimental observations. We investigate the effects of different wall models, subgrid scale models and grid resolution on flow separation by comparing with available data.

Keywords: Computational Fluid Dynamics (CFD), Turbulence, Flow Separation, Large Eddy Simulation (LES).

1 Introduction

Wall-modeled large eddy simulation (WMLES) methods are becoming increasingly popular for high Reynolds number turbulent flow configurations with separation, due to their potential to be more accurate compared to Reynolds-averaged Navier-Stokes simulations (RANS), which are the current industry standard. While the RANS technique models all of the turbulence scales, the WMLES technique resolves a significant portion of the energy-carrying eddies away from the wall. Nevertheless, the current framework and models used in WMLES are far from perfect; further understanding and improvements are necessary to enable their use for complex industrial applications.

2 Problem Statement

Careful CFD validation experiments of flow over a Gaussian bump configuration are being performed by Williams et al. [1] and others to assess the predictive ability of various simulation approaches, i.e., RANS, WMLES and Direct Numerical Simulation (DNS). The experimental conditions were chosen based on close collaboration with experts at Boeing, to be a challenging

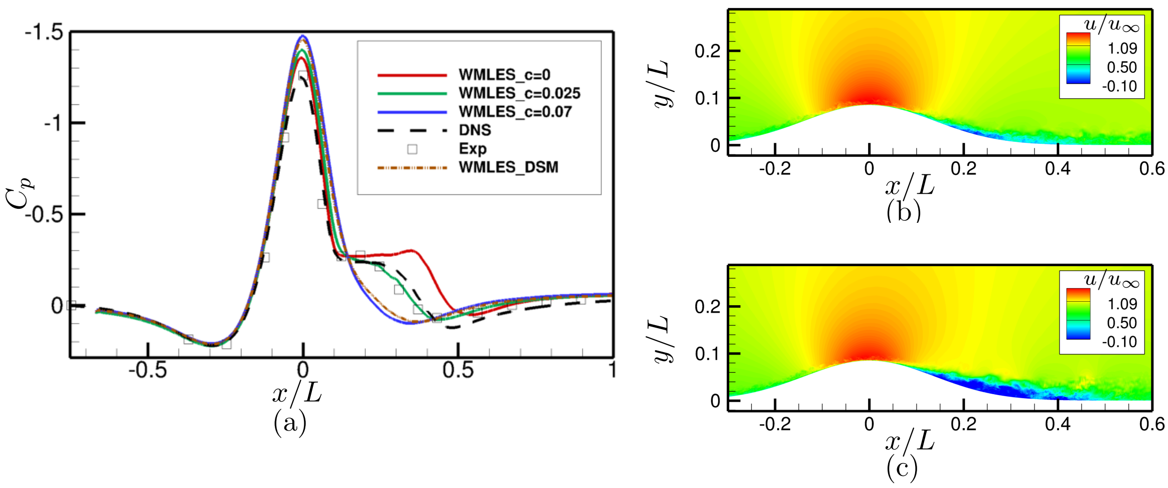

Figure 1: Preliminary WMLES results showing effect of subgrid scale model on (a) the time-averaged wall pressure (C_p), and the instantaneous horizontal velocity contour with the Vreman model coefficient of (b) 0.07 and (c) 0.025.

test case for turbulence modeling approaches. We use the compressible Charles solver from Cascade Technologies*, which uses an unstructured cell-centered finite volume methodology, and an explicit third-order Runge-Kutta time stepping scheme. The equilibrium wall model is used, which imposes the wall shear-stress and heat flux or temperature as a boundary condition, allowing a large near-wall grid spacing. The same framework has been used in previous studies such as flow over a wall-mounted hump, and the NASA juncture flow experiment on an aircraft-type geometry, which has shown promise in predicting separated flows. Our previous study of this problem [2] indicated that while WMLES performed well for the lower $Re_L = 1$ million, its accuracy deteriorated for the higher $Re_L = 3.6$ million where it failed to predict any flow separation, contrary to the experimental observations.

3 Proposed Work

Wall-modeled LES will be performed for the spanwise periodic configuration at $Re_L = 2$ million with different subgrid models (Vreman model with different coefficients, Dynamic Smagorinsky model), wall models (equilibrium and nonequilibrium) and grid resolutions/topology to study its effects on separation. Preliminary results on the effect of subgrid model is shown in Figure 1. Detailed comparisons will be made with available high-fidelity numerical data of Uzun & Malik [3], and experimental data of Williams et al. [1].

References

[1] O. Williams, M. Samuell, E. S. Sarwas, M. Robbins, and A. Ferrante, "Experimental Study of a CFD validation test case for Turbulent Separated Flows." AIAA Paper 2020-0092, 2020.

[2] P. S. Iyer and M. R. Malik, "Wall-modeled les of flow over a gaussian bump." AIAA Paper 2021-1438, 2021.

[3] A. Uzun and M. R. Malik, "High-fidelity simulation of turbulent flow past a gaussian bump," *AIAA Journal*, *https://doi.org/10.2514/1.J060760*, 2021.

*https://www.cascadetechnologies.com, Last accessed: January 18, 2022