COMPARISON OF TURBULENCE AND COMBUSTION MODELS FOR A PREMIXED BLUFF BODY CONFIGURATION

MADHU C. VELLAKAL*, AHMED A. TAHA*, HOSSAM A. EL-ASRAG†, QIYUE LU*, SEID KORIC*§, SHAOPING LI† AND ELLEN MEEKS†

* National Center for Supercomputing Applications (NCSA)
University of Illinois at Urbana-Champaign
1205 W Clark Street, Urbana, IL, USA
e-mail: vcmadhu@illinois.edu, web page: http://industry.ncsa.illinois.edu

† ANSYS INC
10 Cavendish Court
Lebanon, NH, USA

§ Department of Mechanical Science and Engineering
University of Illinois at Urbana-Champaign
Urbana, 61801, IL, USA

Key words: Premixed Combustion, CFD, Bluff Body

Summary. The primary objective of the study is to compare different turbulence-combustion interaction models and turbulence closure approaches for a premixed bluff-body stabilized flame configuration.

1 INTRODUCTION

Bluff bodies are used in various engineering devices to stabilize the flame. Similarly, these bluff bodies are employed in studies to understand the combustion dynamics. The inherent unsteadiness in the domain due to constant interaction of the vortices from the bluff body and the flame surface in the flow field makes it challenging to research configurations both experimentally and computationally. The stabilizing feature of the bluff body can also introduce instability in the flow field. Flames under such conditions are also affected by the turbulence in the flow, which also influences combustion instability in gas turbines. Sjunnesson et.al [1] studied the ‘Volvo Validation Rig’ using physical experiments. A propane-air mixture was combusted in the domain with an equivalence ratio of $\phi \approx 0.65$. The computational domain and the generated mesh for the domain is shown in figure [1]. Various numerical studies have been carried for this configuration. Studies modeling such configurations need to invoke appropriate turbulence models to capture the effects. Large Eddy Simulation (LES) is being used increasingly for engineering high-performance devices like gas turbines. Understanding the complex combustion processes and dynamics is vital to meet growing demands of performance and emission standards. Before deploying the LES approach as the default modeling approach,
there is a need to understand various parameters like reaction mechanisms, sub-grid scale models and combustion modeling approaches and their effect on the ability to predict the physics. Bluff body stabilized flame has been studied experimentally and using LES modeling techniques. The recirculation of the flow behind the bluff body wake is considered necessary in sustaining the flame and the ability to predict the zones is critical in validating the modeling approach. The authors aim to provide an analysis of predictive capabilities of different modeling approaches and their correlation with experimental values for quantities like vorticity, temperature, and velocity.

Figure 1: Computational Domain and Computational Mesh

2 NUMERICAL SIMULATION

ANSYS Fluent, a commercial finite volume based solver, will be used for the simulation. The solver has all the required models to simulate turbulence, heat transfer and reacting flows. Turbulence will be modeled using both the Reynolds-averaged Navier-Stokes (RANS) and the LES method. The length scale of the large eddies can be compared to that of the characteristic length of the mean flow. The dissipation of the turbulence kinetic energy is due to the presence of small scale eddies. Hence it is important to include the effects of both. Resolving smaller eddies to finest scale requires a large computational domain and a significant amount of computing power. For flows with high Reynolds’ number, this becomes very expensive to calculate. RANS simulations model all the different turbulence scales and hence rely on the physical models included in the simulation for the accuracy of the results. LES resolve the large-scale eddies and models the smaller scale eddies. A model fuel library with validated reaction mechanism available with ANSYS will be used. Turbulent combustion simulations must choose between detailed chemistry and fast chemistry and RANS versus LES models while trying to study the combustion processes. Eddy Dissipation Concept model available in ANSYS Fluent models the interaction between turbulence and chemistry using detailed or reduced mechanisms. Similarly, other combustion models like Thickened Flame Model will be investigated as appropriate for the study.

REFERENCES