ORNL/TM-2023/2904
CRADA/NFE-17-06806

Surface Roughness Effects from Additive Manufacturing in High Efficiency Gas Turbine Combustion Systems

Srikanth Allu (ORNL)
Bhaskaran Rathakrishnan (GE Research)

August 2023

FINAL CRADA REPORT (CRADA/NFE-17-06806)

OAK RIDGE National Laboratory

ORNL/TM-2023/2904

High Performance Computing for Manufacturing Final Report

Surface Roughness Effects from Additive Manufacturing in High Efficiency Gas Turbine Combustion Systems

Srikanth Allu (ORNL)
Bhaskaran Rathakrishnan (GE Research)

August 2023

Prepared by
OAK RIDGE NATIONAL LABORATORY
Oak Ridge, TN 37831
managed by
UT-BATTELLE LLC
for the
US DEPARTMENT OF ENERGY
under contract DE-AC05-00OR22725

ABSTRACT

In the past, clean combustion systems were characterized by high system development costs due to growing complexity and escalating manufacturing costs from conventional manufacturing processes. As a result, high efficiency concepts are difficult to design and more difficult to be cost-effectively manufactured. In recent years, with the introduction of additive manufacturing (AM) technologies, the rapid prototyping and mass production processes of clean combustion systems are promising to be significantly simplified with significant reduction in terms of engine manufacturing cost and engine energy cost. Compared with conventional manufacturing built parts, the AM process enabled a simpler design to be adopted for the nozzle, reducing the number of required braze and weld joints from twenty-five to just five. The resulting nozzle was 25% lighter and five times more durable and contributed to a 15% reduction in fuel burn in comparison with the previous model produced. Applying these improvements to a fleet of 5,000 turbofan engines at 150 kN thrust would result in fuel cost savings of $6B annually (at $5/gallon Jet-A fuel price, 700 gallon/hour fuel consumption rate, and 2,300 operational hours per year), and reduce CO2 emissions by more than 25 million tons per year. These advances support core Department of Energy (DOE) missions in energy efficiency, improving productivity, and environmental sustainability. Maximizing the benefit of these new capabilities will require high prediction capability of high speed turbulent flow with wall modeled Large Eddy Simulation (LES). GE Aviation maintains that advanced simulation technology and supercomputing is required in order to provide the appropriate boundary conditions to realize the maximum potential of AM and Ceramic Matrix Composite to reduce cooling flow, a first order penalty on the Brayton cycle.

The algorithm developed using the ANSYS/Fluent software would provide the foundation for entirely new avenues of research and development with potential multi-billion dollar impact to the US economy, and significant reduction in carbon based emissions across the aerospace and power generation industries. The formulation of this iWLES (integral wall model for LES) is generic and allows to capture the changes in flow dynamics that have a significant impact on the wall bounded flow characteristics, such as swirler effective area, bulk swirl number, local pressure distribution, exit velocity profile, and turbulence kinetic energy profile.

A periodic channel flow with rough flat plate is simulated using LES (Wall-Adapting Local Eddy-viscosity) model to verify the implementation of iWLES in the Fluent User Defined Function (UDF). As the flow fields are highly sensitive due to surface roughness of the wall bounded flows in the combustion systems, there is significant potential to conduct further LES studies focusing on the turbulence boundary layer interaction. Accurately capturing near wall physics will elucidate the impact of rough surfaces on combustor flow and aero-thermal interactions.

1. INTRODUCTION

Additive manufacturing has been increasingly adopted by gas turbine manufacturers for rapid prototyping and mass production of complex, light-weight parts for combustion systems. However, the AM process typically results in higher surface roughness compared to conventional manufacturing. Surface roughness effect, which is difficult to be efficiently modeled with high accuracy, results in performance difference between drawing board concepts and as-built parts, which poses challenges on combustion system efficiency improvement and manufacturing cost reduction. This challenge potentially results in significant differences in aerodynamics characteristics and combustion processes, which depends on the realistic surface boundary conditions of mixers, such as surface roughness. Surface roughness effect, compared to an "as designed" ideally clean surface, is a major factor for aerodynamic characteristics deviation of manufactured hardware compared to concepts on the drawing board. Surface roughness results in small scales of perturbation that contribute to the early development of turbulence and boundary layer separation. In gas turbine applications, the changes in flow dynamics have a significant impact on the wall bounded flow characteristics, such as swirler effective area, bulk swirl number, local pressure distribution, exit velocity profile, and turbulence kinetic energy profile. With new manufacturing methods such as AM without post machining, the effect of raw surface roughness, in certain ranges of lengths scales is very difficult to understand and predict. As of today, most combustion computational fluid dynamics boundary conditions are not well quantified in terms of surface roughness. To conduct design concept simulation, the surface roughness is either ignored or estimated based on manufacturing material and surface treatment process, which is considered as empirical and insensitive to local qualities. This new wall modeled LES will provide assessment of surface roughness impact on high speed flows in gas turbine combustor, and how to efficiently and effectively model this effect using a new generation of wall models, for a well-controlled gas turbine test facility with high quality measurement data. This will provide opportunity to enable the design and manufacture of advanced gas turbine combustors with higher pretest prediction capabilities, a better patterning of combustor exit profile to reduce cooling flow supplied to the high pressure turbine, and directly improving fuel efficiency of the engine and prolonging life of critical high temperature components.

A new integral wall model for LES (iWLES) is implemented as a user defined function (UDF) in massive parallel commercial software ANSYS Fluent. To properly include near-wall physics while preserving the basic economy of equilibrium-type wall models, we adopt the classical integral method of von Karman and Pohlhausen (VKP). A velocity profile with various parameters is proposed as an alternative to numerical integration of the boundary layer equations in the near-wall zone. The profile contains a viscous or roughness sublayer and a logarithmic layer with an additional linear term that can account for inertial and pressure gradient effects. Similar to the VKP method, the assumed velocity profile coefficients are determined from appropriate matching conditions and physical constraints. Next, we describe the simple functional forms for turbulent boundary layer profiles that were used in the model.

The assumed velocity profile has the form:

$$\langle u \rangle = u_\nu \frac{y}{\delta_\nu} \quad 0 \le y \le \delta_i$$

$$\langle u \rangle = u_\tau \left[C + \frac{1}{\kappa} log \frac{y}{\Delta_y} \right] + u_\tau A \frac{y}{\Delta_y} \quad \delta_i \le y \le \Delta_y$$

The parameters in these profiles $(C, A, \delta_i, \delta_\nu, u_\nu, u_\tau)$ are solved for using 6 physical constraints given below:

$$u_\tau (C + A) = U_{LES}$$

$$u_\nu \frac{\delta_i}{\delta_\nu} = u_\tau \left[C + \frac{1}{\kappa} log \frac{\delta_i}{\Delta_y} + A \frac{\delta_i}{\Delta_y} \right]$$

$$\delta_i = min \left[\max \left(k, 11 \frac{\nu}{u_\tau} \right), \Delta_y \right]$$

$$\delta_\nu = \frac{1}{u_\nu} (\nu + \nu_{T,y=0})$$

$$\tau_w = u_\tau^2 = u_\nu^2 + \int_0^k C_d a_L \langle u \rangle^2 dy$$

$$\frac{dL_x}{dt} + M_x = \tau_{\Delta_y} - \tau_w = \left(\nu + \nu_T|_{y=\Delta_y} \right) \frac{u_\tau}{\Delta_y} \left(\frac{1}{\kappa} + A \right) - u_\tau^2$$

These coupled equations are solved numerically using Newton iteration at every grid point along the wall boundary.

2. RESULTS

Surface roughness results in small scales of perturbation that contribute to the early development of turbulence and boundary layer separation. A new wall model is implemented as a user defined function in massive parallel simulation within ANSYS Fluent software. The formulation of this iWLES (integral wall model for LES) is derived for an arbitrary orientation of the wall coordinate system. This new model allows to capture the changes in flow dynamics that have a significant impact on the wall bounded flow characteristics, such as swirler effective area, bulk swirl number, local pressure distribution, exit velocity profile, and turbulence kinetic energy profile. A periodic channel flow with LES (WALE) model is employed to verify the specification of iWLES as wall model. The mesh used in the simulation has a total of 216,000 cells and the total time of simulation corresponds to 1,200 flow through times. In this setup we verify the specification of wall shear stress profile as boundary condition. The built-in version on top wall and the UDF version of the same function on the bottom wall produces nice and symmetric solution. This shear stress UDF is utilized as baseline for implementation of iWLES model. Figure 1 shows the comparison of the weighted average wall shear stress between the no slip and iWLES imposed boundaries computed over 100 seconds. This shows that the wall model implemented as UDF is behaving as expected and we see minor variations in the wall shear stress that could diverge if we continue the computations further. To see if this is a potential issue that needs to be address, we identified a second challenge problem with prescribed surface roughness and has wall resolved solutions and metrics available in the research literature.

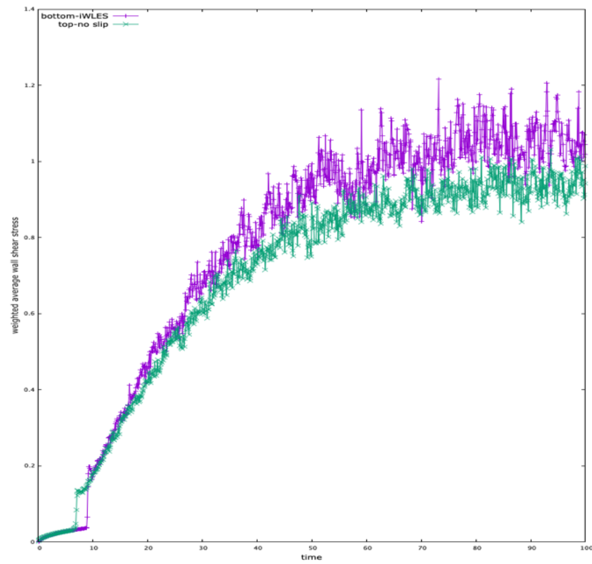
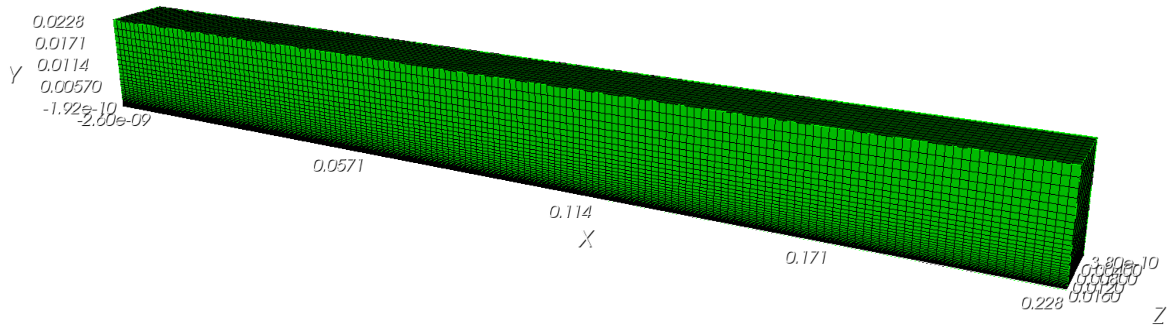


Figure 1 Comparison of weighted average wall shear stress between no slip and iWLES imposed boundaries.

Next, a new benchmark is studied with boundary layer developing over the plate with a sharp leading edge. To investigate the effect of distributed roughness on subsonic boundary layers the rough surface is specified over the entire length of the flat plate. The numerical setup and analysis that is carried to compare the transition of subsonic boundary layer on a flat plate with roughness elements distributed over entire surface. The problem geometry consists of a flat plate with roughness elements distributed along the streamwise and spanwise directions. The flat plate extends to $L_x = 10000\theta_0, L_y = 1000\theta_0, L_z = 700\theta_0$ in streamwise, wall-normal and spanwise directions where θ_0 is the momentum thickness of boundary layer at the leading edge of flat plate. The momentum boundary layer thickness is calculated to be 2.314×10^{-5} with the free stream Mach number is set to 0.5. Periodic boundary conditions are employed in the spanwise directions. Reynolds Averaged Navier-Stokes (RANS) simulations of the flat plate was conducted as an initial baseline study. The time step for LES simulation is chosen to be $0.1\theta_0/U_\infty$ and the computations were conducted over a couple of flow through times. The objective of this project is to study the impact of the roughness elements on the turbulent boundary layer and for the chosen roughness characteristics we use drag coefficient $C_d = 0.1$ and roughness areal density $a_L(y) = 1$ as parameters in the iWLES model.



Sample Mesh used for flat plate benchmark simulation

The height of the roughness elements/features corresponding to the boundary layer will generate an elevated shear layer and alternating high and low speed streaks near the wall. The comparison of onset of flow transition predicted with the iWMLES simulation with roughness resolved LES will give an assessment of the proposed approach.

The baseline simulations were needed to facilitate assessment of mesh resolution and the improvements required for the LES analysis. Furthermore, the RANS simulations were needed to provide initial conditions in the form velocity profiles for the LES simulations. With the given free stream boundary condition, we determined the skin friction coefficient to be $C_f = 0.0045378$. For a $y^+$of 50 we end up with the mesh where the first cell adjacent to the wall is at the distance 9.18095×10^{-5}. The numerical investigations are conducted on this channel mesh with free-stream boundary conditions applied on the top boundary and periodic boundary conditions are employed in the spanwise direction. The effect of surface roughness on transition is studied and comparisons are made against the transition onset correlations. Figure 2 shows the comparison of velocity profiles and wall y^+ with increased Reynolds number.

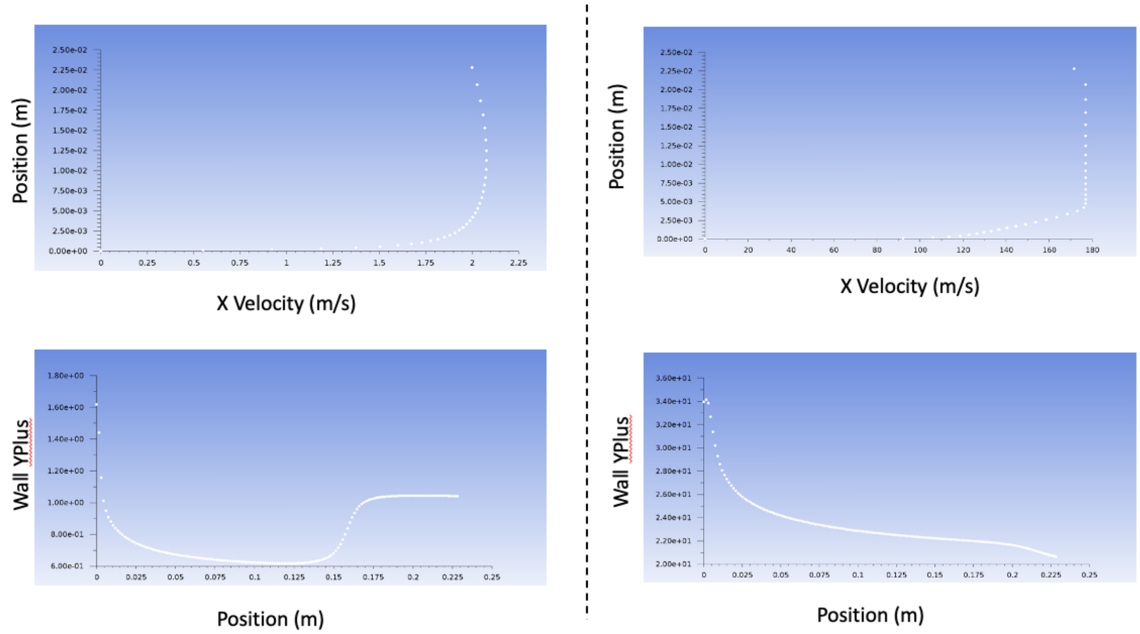


Figure 2 Comparison of velocity profiles and wall y^+ with increased Reynolds number

The RANS simulations were conducted until a steady state solution was reached. The LES simulations are currently being conducted over a multiple flow through times and are still pending completion.

3. IMPLEMENTATION

The project commenced after acquiring the high performance computing resources and subsequently ANSYS/Fluent licenses were installed on leadership computing facilities. As the original PI was displaced due to reorg, the task to create UDF for the Fluent software implementing the integrated wall model for LES computations was taken over by the Oak Ridge National Laboratory team. Next, we performed work on channel flow mesh to confirm performance of the implemented wall models using UDF as test problems. Finally, we conducted cold flow simulations on leadership computing facilities of the baseline RANS/LES cases along with the provided iWMLES boundary model using Fluent. The final task of reacting flow simulation was not performed as the pandemic has presented its own set of challenges at both organizations participating in the project.

4. CHALLENGES

There were many unexpected challenges that the team encountered during project execution. The first delay in starting the project was due to multiple administrative reasons, including a reorg within the industrial partner institution, change of PI, and paperwork with Ansys to have Fluent licenses in place for the project. Next, halfway through the project the ASCR Leadership Computing Challenge allocation came to an end and was not renewed at the Oak Ridge Leadership Computing Facility causing significant delay. This was addressed by procuring the required HPC resources at National Energy Research Scientific Computing Center (NERSC). But the issue was complicated further with the ANSYS licenses that needed to be installed on new architecture, i.e., on NERSC machine. Finally, with the help of support staff at NERSC we were able to test and launch the ANSYS software which paved the way for the development/implementation of the model required for the project.

As the flow fields are highly sensitive due to surface roughness of the wall bounded flows in the combustion systems, there is significant potential to conduct further LES studies focusing on the turbulence boundary layer interaction. Accurately capturing near wall physics will elucidate the impact of rough surfaces on interactions between pressure gradients and highly complex turbulent features. The wall models used along with the LES can be improved further in combination with combustion models to make the simulations predictive.

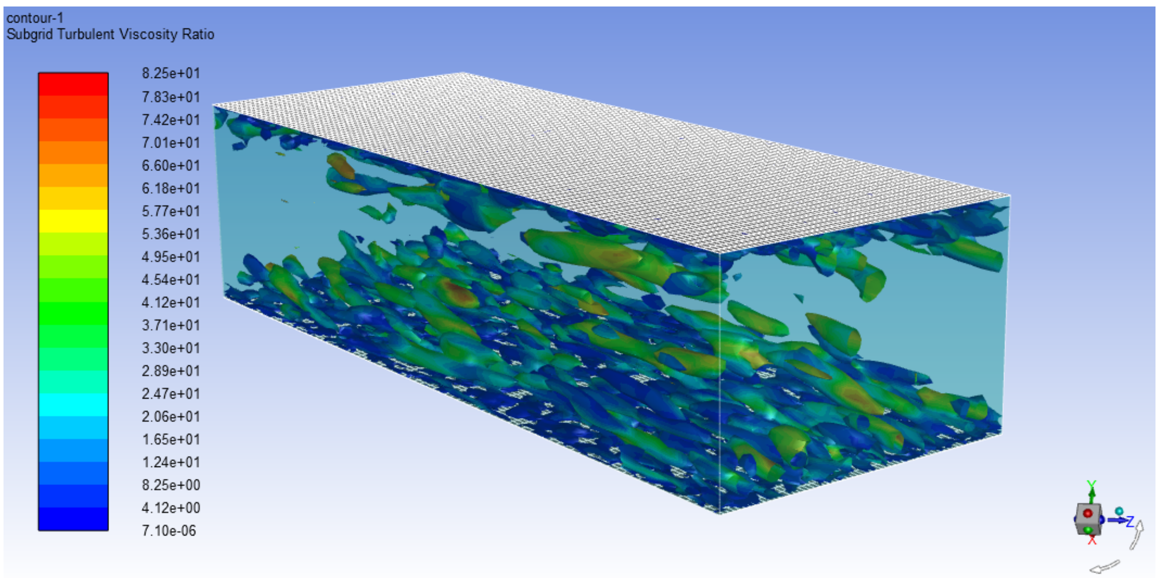
5. IMPACT

Lean burn liquid fuel combustion systems, where fuel and air are premixed before entering the gas turbine combustor, are emerging as a preferred technological solution for fuel efficiency, operational flexibility, and low emissions in power and propulsion applications. For example, advanced lean burn technologies are an important element of recent Federal Aviation Administration initiative of gas turbine programs that promise to reduce aircraft fuel burn by 40%, the emission target by 2025 set by International Civil Aviation Organization. Applying these improvements to a fleet of 5,000 turbofan engines would result in fuel cost savings of $6B annually (at $7.5/Gal Jet-A price and 2,700 operational hours per year) and reduce CO2 emissions by more than 25 million tons per year. These advances support core DOE missions in efficiency and environmental sustainability.

GE Aviation is investing in both advanced AM and digital twins of aircraft engines to improve engine fuel efficiency and operating efficiency from improved durability. Maximizing the benefit of these new capabilities will require high prediction capability of high-speed turbulent flow with wall modeled LES. The developed models would provide the foundation for entirely new avenues of research and development with potential multi-billion dollar impact to the US economy and significant reduction in carbon based emissions across the aerospace and power generation industries. Furthermore, this effort will provide an opportunity to enable a major US manufacturing company to remain competitive with the EU in the context of the significant FP-7 FACTOR project currently being executed.

APPENDIX A. Additional Results

As the flow fields are highly sensitive due to surface roughness of the wall bounded flows in the combustion systems, there is significant potential to conduct further LES studies focusing on the turbulence boundary layer interaction. Accurately capturing near wall physics will elucidate the impact of rough surfaces on interactions between pressure gradients and highly complex turbulent features. The wall models used along with the LES can be improved further in combination with combustion models to make the simulations predictive.



Iso-surfaces of Q criterion indicating presence of turbulent structures.