Combustion Laboratory Research
Department of Mechanical Engineering
July 3, 2019, Statesboro, Georgia, USA

Research Update Report

Simulated Flow and Combustion Analysis of Gas Engine Combustion Systems

Drake T. Grall[1]
Georgia Southern University, Statesboro, GA, 30458, USA

Dr. Valentin Soloiu
Supervising professor, Georgia Southern University, Statesboro, GA, 30458, USA

Dr. Marcel Ilie
Supervising professor, Georgia Southern University, Statesboro, GA, 30458, USA

I. SR-30 Turbojet Engine

The SR30 Turbojet Engine is a research-focused gas-turbine that can operate on multiple different fuels derived through various sources. The high flexibility of the engine lends too investigation of engine efficiencies and performance under different operating conditions. The purpose of the research presented in this paper is to develop 2D mathematical and 3D Computational Fluid Dynamics (CFD) simulation models capable of predicting, within a reasonable level of certainty, the efficiency and operating performance of the engine under varying operating conditions (i.e. different fuels, engine speeds, operating environment, etc.). The models are validated with experimental data to test the accuracy and robustness. Primary motivating factors for this research are lessening the financial cost associated with operating the engine, reducing risk to engine as well as operators when experimenting with different engine operating conditions, providing another means for validation of future research topics involving the SR30 Turbojet Engine, developing a general framework and methodology for others to implement this modelling method with other engines, and to further develop this area of research in the aerospace and simulation communities.

A. Preliminary Flow Analysis Results

1. Velocity

1 Research Assistant, Department of Mechanical Engineering.

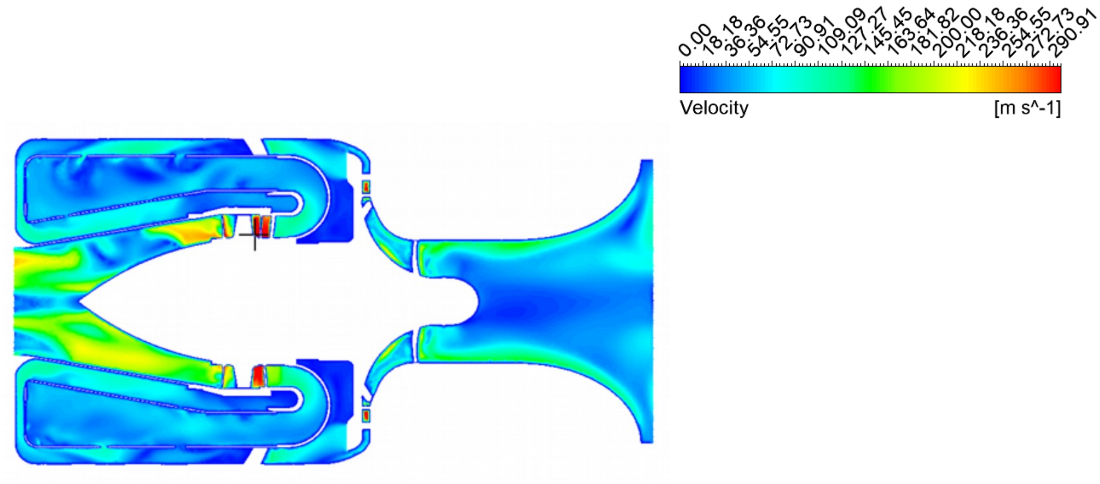

Fig. 2 m= 0.5 kg/s Velocity

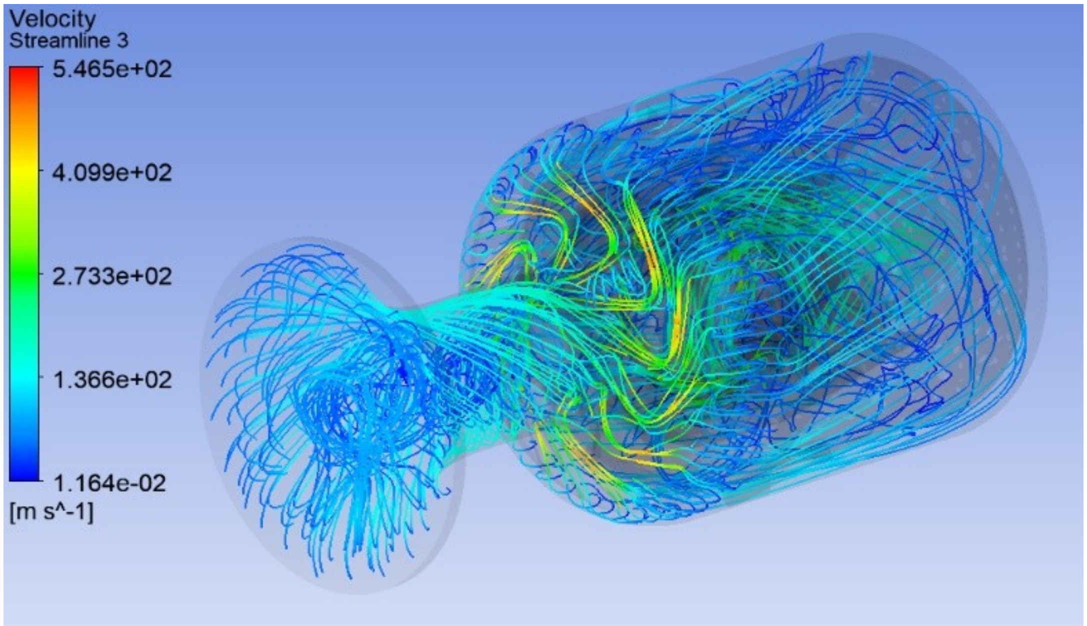

Fig. 1 Velocity Streamline Iso View

The preliminary results for this topic yielded results that match closely with values and specifications provided by the manufacturer as well as experimental operation data. At a peak operational design point of 90,000rpm, the engine consumes on average 0.5kg/s of air and fuel with almost 96% of this mass flow rate consisting of air. At these operating conditions the maximum velocity of the air flowing through the engine, based on manufacturers specification, should be 473m/s at the tip of the centrifugal impeller blades. Above in Figures 1 and 2 the velocity streamlines and the velocity flow are displayed. This model displays peak velocity at the tips of the compressor blades and moderate to high velocity at the tips of the turbine blades. The mathematical and CFD simulation models produced air velocities results at the tip of the centrifugal impeller blades of 494.6m/s and 478m/s respectively. These results and the ongoing developments of this work will require the addition of dynamic and combustion models to further predict and study performance.

2. Vorticity

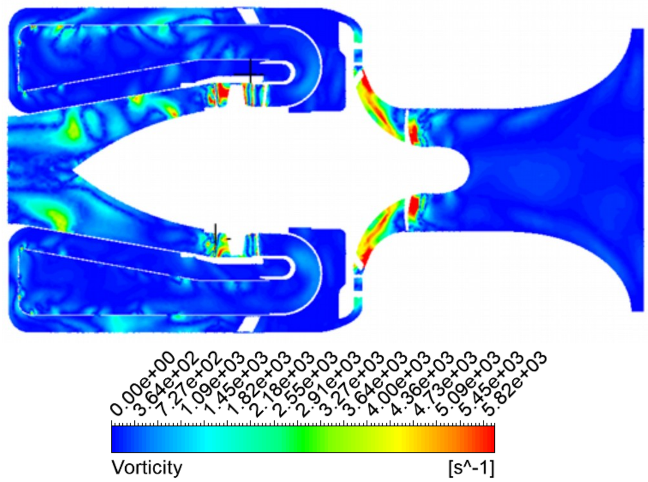

Fig. 3 . m= 0.5 kg/s Vorticity

The vorticity of a fluid flow is the local spinning, or rotating, motion of particles at a given point. Vorticity is calculated based on the relative position of a particle and the relative displacement through the flow in a localized region. The vorticity in the simulation, shown in Fig. 3, is observed, presented in a logarithmic scale, and is seen to be greatest after the compressor blades and the axial turbine blades. The small rotating motion of the air molecules after each component increases the eddy swirl.

II. Gas Turbine Can Combustor

This study is dedicated to manipulating the combustion reactions within a commercial style can combustor. This simulation utilized Eddy dissipation combustion and the P1 radiation model to simulate the combustion of gaseous methane within the can combustor. Studying the behavior of different combustion models allows for a more accurate approximation of performance and behavior of varying fuel types.

A. Preliminary Results

1. Temperature

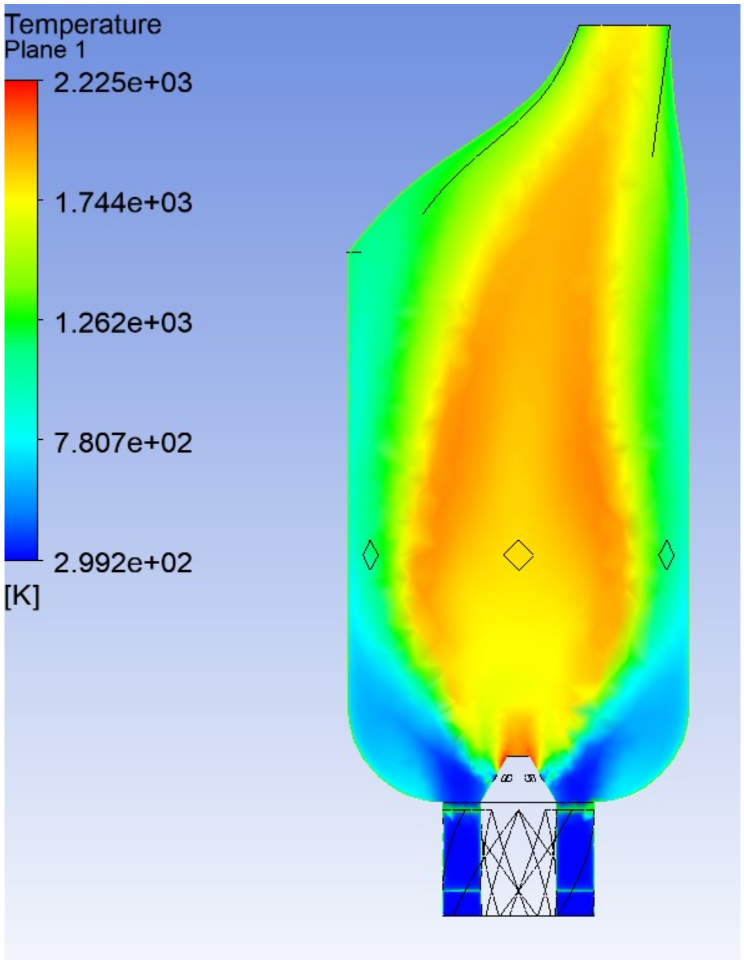

Fig. 4 Total Temperature of Can Combustor

As you can see in Figure 4 the total temperature of the methane combustion the flame propagation has higher temperatures near the injector where combustion begins. Also, the flow of the heat is consistent with the expected behavior flowing out and creating large amounts of heat and energy as the flame exits.

2. No Mass Fraction

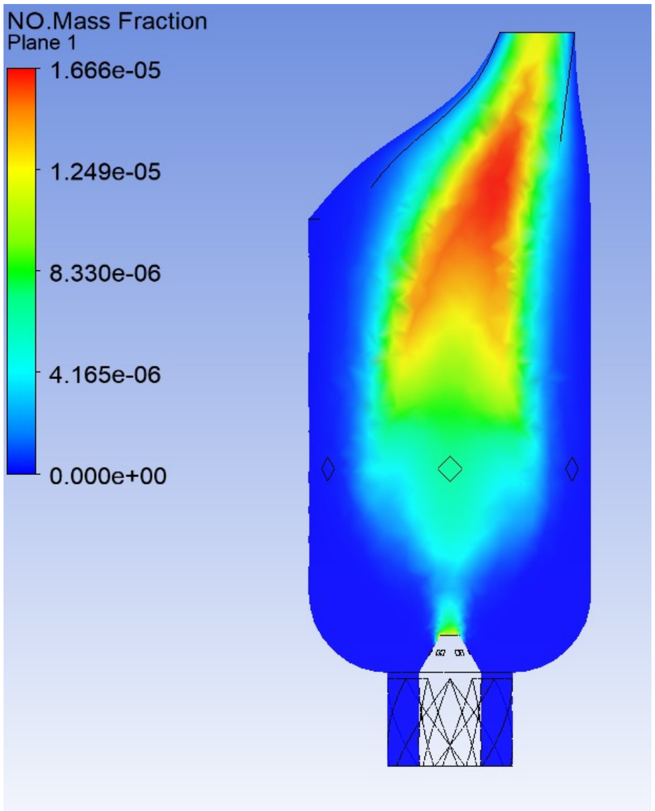

Fig. 5 NO Mass Fraction of Methane Combustion

Figure 5 displays the Nitric Oxide (NO) mass fraction which is a byproduct of combusting fuels with air creating exhaust. In this case the peak creation of NO forms more closely to the outlet and center of the can combustor where the combustion is occurring and is well within acceptable percentages of 0.0000166 parts per million (ppm). The volume rendering shown in Figure 6 is give a better view of how the NO mass fraction develops as the combustion begins. It is recommended that for further study varying fuel types and properties be introduced to comparatively study the reactions that can develop with combustion.

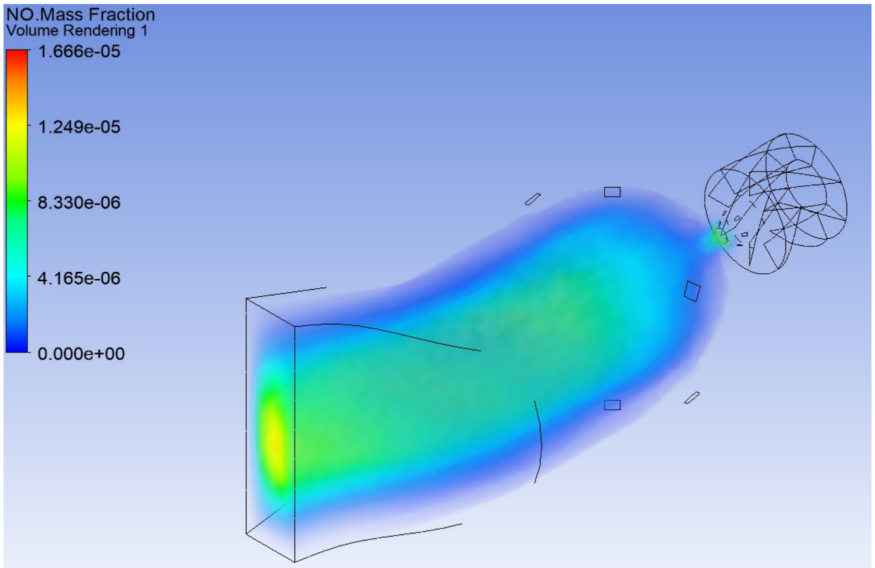

Fig. 6 NO Mass Fraction of Methane Combustion

III. Yanmar Spray Injection Analysis

This study developed by fellow researchers David Mothershed, Justin Wiley, Tiesha Wolfe, and Angelika Quedraogo sought to find methods for emission reduction through spray pattern optimization for a Yanmar TS230R direct injection research engine. Emission control is an important topic of research and development in order to not only prevent the release of harmful toxins into the environment, but also produce better and more efficient combustion processes.

A Volume Fraction Preliminary Results

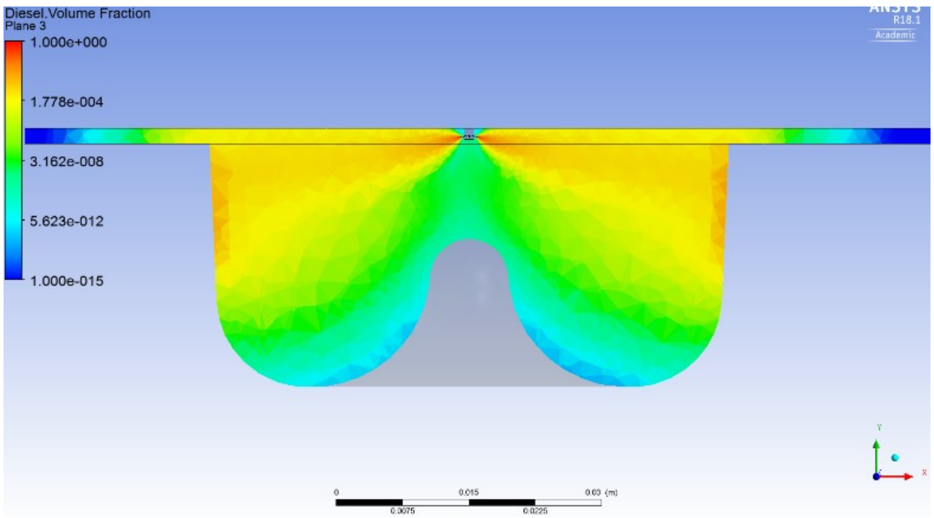

Fig. 7 NO Mass Fraction of Methane Combustion

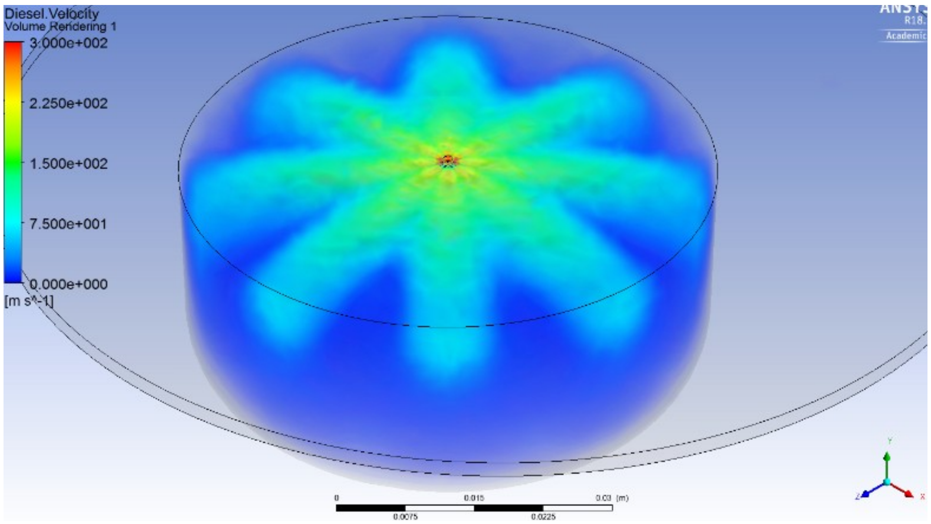

Fig. 8 NO Mass Fraction of Methane Combustion

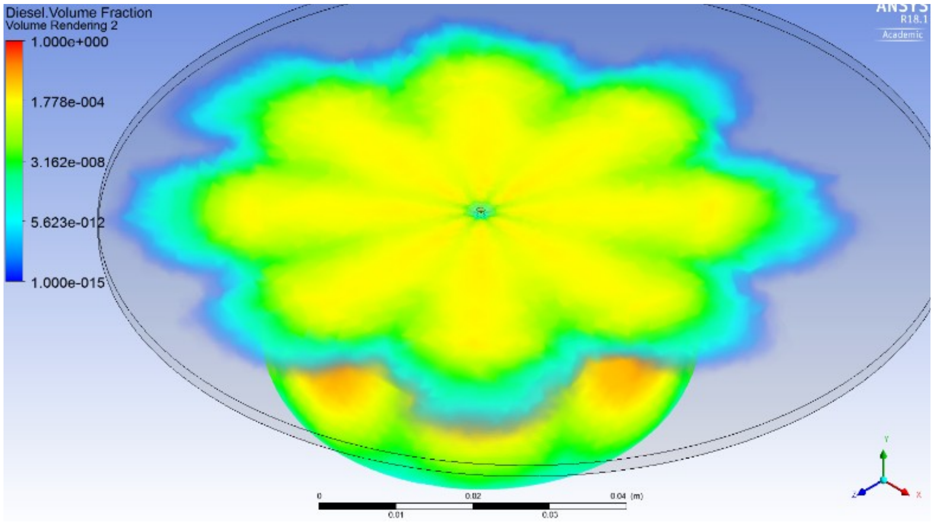

Fig. 9 NO Mass Fraction of Methane Combustion

The three different spray patterns simulated were a single straight spray, a four spray with a 170° spray angle, and an eight spray with a 155° spray angle. Through this study it was shown that the eight-spray injector provided greater atomization of fuel with a more even spray distribution which can be observed in Figures 7-9. It also showed greater spray penetration into the cylinder of 52mm from the injector tip compared to the 45mm penetration of the four-spray simulation. Using the established methods in this simulation more intricated spray models can be developed and applied to a variety of different pieces of equipment and models. Moving forward a concentration on developing fuel reaction and materials for simulation will be necessary in order to test other forms of fuels.

IV. Cetane ID 510 Constant Volume Combustion Chamber Analysis

This study is dedicated to model and simulating various forms of combustion using the PAC Cetane ID 510 Constant Volume Combustion Chamber (CVCC) using a variety of different fuel and fuel characteristics. Due to some product specification being limited a single inject port was utilized to study the spray atomization within the CVCC as the fuel combusted. The fuel utilized in this preliminary study was a form Diesel also referred to as Decane.

B Preliminary Results

1. Velocity

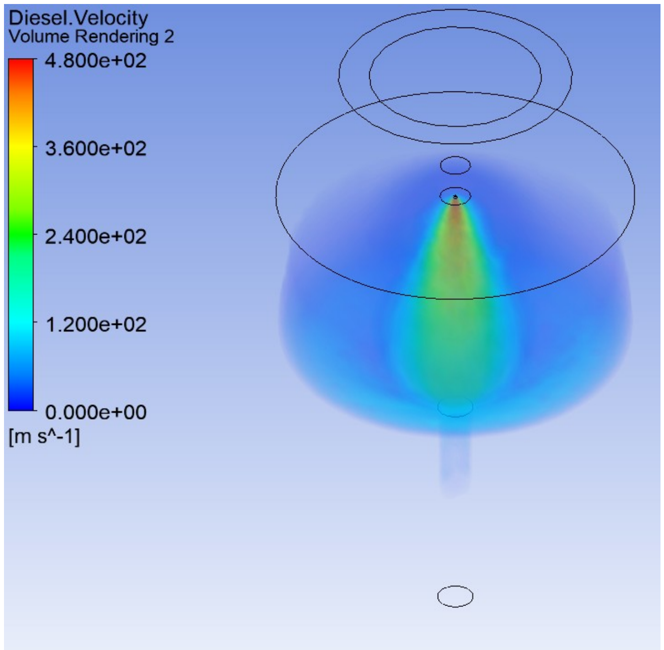

Fig. 10 Single Spray Diesel Velocity Volume Render Iso View

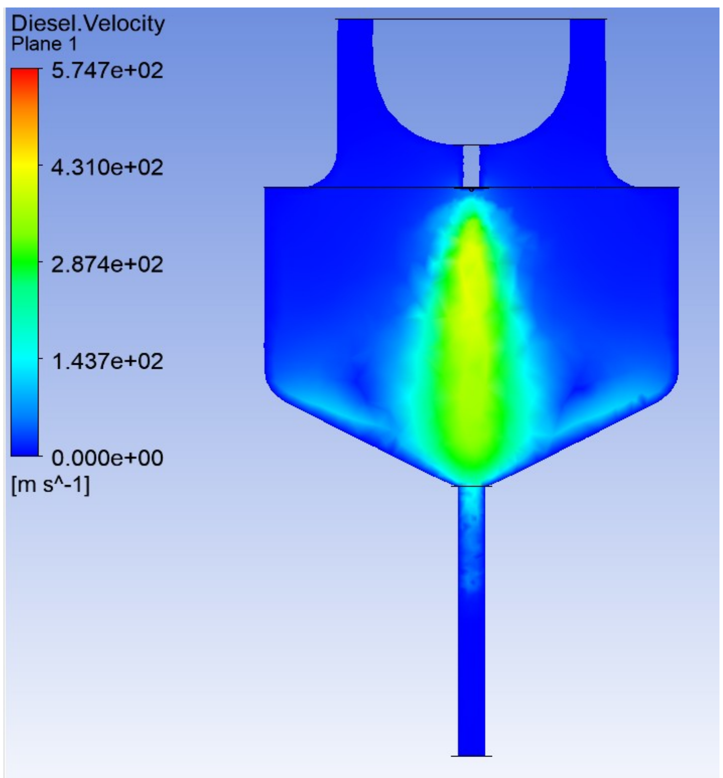

Fig. 11 Single Spray Diesel Velocity Side View

In Figure 10 the diesel velocity volume rendering can be observed and as theorized the high rate of velocity at 480 m/s^2 occurs at the begin as the fuel is injected and begins to combust. This indicates that the fuel characteristics are appropriate and that the study is operating correctly. Figure 11 shows the side view of the diesel velocity as it is injected into the CVCC.

2. Diesel Volume Fraction

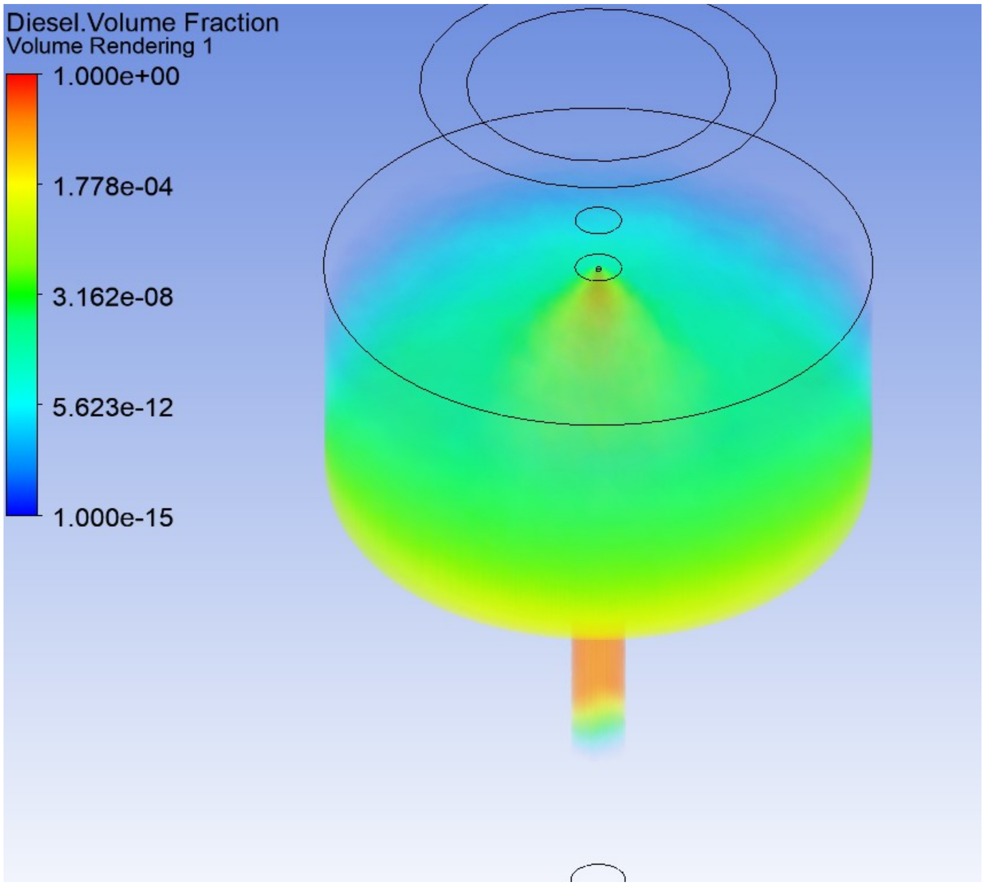

Fig. 11 Single Spray Diesel Volume Fraction Volume Render Iso View

Figure 12 shows the volume rendering of the Diesel Volume Fraction apparent in the combustion model. Larger amounts of time and concentration of fuel characteristics will be required in order to more accurately model the combustion behavior inside of the PAC Cetane ID510 CVCC. The next step required will be to adjust the injector model to have a more accurate spray analysis with the correct orifice sizing and spacing required.