**ABSTRACT**:

Primary purpose of this study is to validate the two test cases viz. Sandia national laboratory propane jet and Sydney bluff body jet flame against the turbulence models available in ANSYS CFX 16.1 and ANSYS Fluent 17.0. This study is divided in to two sections a) Investigation of Sandia propane jet b) Investigation of Sydney Bluff body jet. Both investigations involve simulations carried out under isothermal, non-reacting and steady state conditions. Sandia propane jet configuration involves quasi-2D meshes, defined as the 2D mesh rotated by 5° in the circumferential direction, used for computation. In order to validate the test configurations numerically the turbulence models k-ω SST, k-ε, BSL RSM (Base Line Reynolds Stress Model) and EARSM (Explicit Algebraic Reynolds Stress Model) from ANSYS CFX 16.1 and k-ω SST, standard k-ε, realizable k-ε models from ANSYS Fluent 17.0 were used. Round jet anomaly phenomenon was addressed and modification in the dissipation equation of k-ε model was applied and the results compared with the experimental data. In the second case study with Sydney Bluff Body Jet flame involved full 3D meshes for the numerical investigation. Investigation was carried out with k-ω SST and k-ε turbulence models in ANSYS CFX 16.1 and k-ω SST, standard k-ε and realizable k-ε turbulence model in ANSYS Fluent 17.0.

In the Sandia Propane Jet case, a small single vortex recirculation region was obtained behind the bluff body. Centerline jet velocity in case of mean axial velocity (U m/s) component, compared with the experimental data, was under-predicted by all RANS models used for this investigation. The reason for this can be the simplifying assumptions made in the RANS models i.e. isotropic turbulence and round jet anomaly phenomenon. Results from modification in the constant delivered that, on one side improvement is obtained in the results of certain quantity and on other side there is deterioration in the result of other quantity. Difficulties in order to achieve convergence were observed in the k-ε turbulence models, but they delivered results that were in good agreement with the experimental data along with k-ω SST model.

In the Sydney Bluff Body jet case, a double vortex recirculation region is obtained behind the bluff body. Three mixing layers are observed in the recirculation zone. It was found that the smaller vortex near the jet loses the circulation pattern with the increase in the momentum of jet fuel. It was observed from the comparison of the mixture fraction profile with the experimental data that k-ω SST turbulence model, in ANSYS CFX 16.1, over-predicts the mixing slightly in certain locations of the domain. Realizable k-ε model, in ANSYS Fluent 17.0, over-predicted the mixing significantly, in certain domain locations, as observed from the mixture fraction comparison plots. Apart from that other turbulence models were in good agreement with each other as well as the experimental data.