論文の内容の要旨

Numerical Simulations on Combustion Characteristics in a Pre-Cooled Turbo Jet Afterburner

（予冷ターボジェットエンジンアフターバーナーの燃焼特性に関する数値計算）

ヤヌシ　ジョージ

A study to investigate the combustion characteristics on hydrogen angled injection has been performed in a model afterburner of the Pre-Cooled Turbo Jet engine (PCTJ). Fuel-lean and fuel-rich simulations have been performed at different injection configurations to evaluate the chemical heat release, total heat loss, combustion efficiency and NOx production. The study is useful to provide good understanding on the flow behavior and combustion characteristics that will later be used in the experimental evaluation of the PCTJ combustor. Computational Fluid Dynamics (CFD) simulations are run in a steady state manner solving the Reynolds Averaged Navier-Stokes (RANS) conservation equations augmented by a two-layer k-ε turbulence model using the SIMPLEST algorithm, over a three dimensional domain (3D). The domain boundary conditions isothermal upper and lower side of the axial flow and periodic on the left and right sides of the axial flow. The chemical interactions taking place in the combustion process were calculated using the CHEMKIN sub-routine. The chemical reaction mechanism used in this simulation involved 188 elementary reactions and 28 chemical compounds. The detailed NO chemical reaction system has been selected to provide a detail understanding of the NOx emission in the combustion process. The angle configurations used throughout the calculations for all equivalence ratios range from 10° to 120°. Data for different configurations is plotted at a constant equivalence ratio (0.26 – lean and 2.0 – rich) when one pair of angles is kept constant and the other varied, with the process repeated for the other case respectively. The combustion efficiency is evaluated using two approaches, (A) uses an enthalpy difference between the outlet and inlet enthalpies in relation to hydrogen theoretical heat release. This first approach is used to have a comparison basis between experiments and numerical simulations. The (B) second approach evaluates the combustion efficiency as the ratio between the chemical heat release and the theoretical heat release. The heat loss at the walls is investigated in the entire chamber. The combustion efficiency is investigated near the

nozzle area for comparison with experiments, and throughout the combustor to determine the flow influence induced by the angles injection. The EINOx is investigated at the exit of the combustor for different injection configurations. Experiments have shown a large temperature difference between the injection region, around the injector, and the nozzle region. As the equivalence ratio increased, the temperature difference decreased. This phenomenon has been investigated numerically, and the cause was determined to be due to the flow behavior induced by various angles of injection and heat loss incurred during the combustion process. Experiments were performed using a high enthalpy wind tunnel on model combustor with one injector. Using a system of thermocouples and static pressure sensors, relevant data was recorded for a fixed fuel-lean and fuel-rich equivalence ratio.

Simulations at fuel-lean conditions have found a that constant angled injection on one pair (upstream or downstream) coupled with increasing angles on the other pair, lead to higher combustion efficiency, when it is evaluated as an enthalpy difference. Higher angles on the downstream increase the mixing in the area and as a result, the temperature in the area. With increasing mixing, and a higher temperature, the NOx emission increase. However, as a result of higher angles, the heat loss at the walls also increased. This lead to the evaluation of the combustion efficiency using the chemical heat release approach, near the injector. The second approach is fairly independent of the flame temperature and heat loss in the combustor. Following this approach, we find that combustion efficiency is higher for increasing angles and lower when angles inject fuel perpendicular and up-flow.

For fuel-rich combustion the performance is substantially influenced by upstream angles due to the vortices that form in front of the injector at high angles lowering the maximum temperature of the flame. Another factor is the presence of the recirculation zones in the wake of the injector, which is found to lower the combustion efficiency. As the mixture becomes richer the combustion zone moves closer to the injector producing a smaller amount of NOx in the exhaust gas. Simulations at fuel-rich conditions show the heat loss is influenced by the mixture’s prolonged contact with the walls, and by vortices forming between the two pairs of injection holes. The chemical heat release was influenced by the size of the vortices in the wake of the injector which affected the velocity flow field. The large vortices that form in the wake of the injector break down into smaller formations, when large angles inject fuel closer to the walls and are reflected, increasing turbulence and combustion efficiency, and heat loss at the walls. It has been found that NO is created in the post-flame region and increases downstream in the combustor. The concentration of NO2 decreased significantly in the wake of the injector, in the post-flame region, and was coincident with a corresponding formation of NO.

A sensitivity study has also been performed to determine the important reactions that create NOx in the current combustor. NO is produced in the center of the high temperature values of the flame through the Zel'dovich mechanism, NO2 recombination and to a smaller degree, by the N2O destruction. The NO2 production is more pronounced in large vortices that form as a result of angled injection. The NNH route is found to have a small influence on the creation of NO through re-dissociation into NO precursors.