Recent Research Activities,
P. Koutmos, LAT

Hybrid/Dual Fuel multi-port Injected Combustor (HDFIC) arrangement

Driven by regulation and environmental concerns, the exploitation of ultra-lean combustion has emerged as promising technology to control emissions from practical combustion. Ultra-lean flame configurations formed by premixing of fuel and air and stabilized on a Hybrid/Dual Fuel multi-port Injected Combustor (HDFIC) arrangement are investigated both experimentally and computationally. Multiport dual fuel injection is exploited to promote flame stability and reduced emissions.

A hybrid/dual fuel injection operation is expected to achieve and maintain stable operation at limiting ultra-lean mixtures. The choice of optimum injection placement/fuel combination configurations and their verification is the core objective of the work. The emission performance of these configurations is compared against standard injection methodologies. Experiments have been performed in the medium scale combustion facility employing: a) high temperature fast response thermocouples, coupled to a DaqTemp 7A Omega card, b) exhaust gas analysis, c) CCD camera recordings of flame images and d) Laser Velocimetry to assess the momentum fields of the flames.


Experimental and computational investigations of axisymmetric buoyant flames and fires.

The objective of these concerted efforts on buoyant flames, in cooperation with local authorities, is to improve understanding of the mechanisms that control open or enclosed fire configurations with the aim to achieve successful models to mitigate their effects. Both experiments and simulations are exploited in a series of studies related to water mist suppression of axisymmetric fires, identification of line fires base characteristics, control of the onset of fire whirls and aerial (water or chemicals) bombardment of wildland forest or spill fires.


Study of large scale vortex dynamics in square cylinder reacting wakes under co-current or counter-current fuel injection.

Current designs of industrial burners usually incorporate bluff-body nozzles to improve flame stabilization, increased efficiency and reduced pollutant emissions. A challenge in bluff-body turbulent combustion modelling is the influence of the large scale
flow structures and the time-varying flow behaviour on flame characteristics such as stability, heat release and emissions. Turbulent flames stabilized by planar propane injection across the span of a slender square cylinder (discrete jets of small aspect ratio), either from its leading face against the approach cross-flow or directly within its vortex formation region are studied. Cold flow studies, turbulent temperature measurements, exhaust gas analysis and reacting Large Eddy Simulations, undertaken for two Fuel Air Velocity Ratios (FAVR) of 0.3 and 0.2 at a Re number of 8000 described the dynamic development of cold and hot wakes under counter- or co-current fuel injection.

A study of the interaction of swirl flow with annular partially premixed propane flames.

The present project deals with the experimental and computational investigation of turbulent reacting wakes established through staged fuel-air premixing in an axisymmetric double cavity arrangement formed along three concentric disks and stabilized in the downstream vortex region of the afterbody. The innovative burner assembly is also operated with a swirling coflow, introduced upstream of the burner exit to allow for the interaction between the primary premixed recirculating afterbody flame and the swirl. The isothermal interaction of the cavity produced annular jet stabilized by the afterbody and the variable swirl is initially studied. Ultra-lean flames with strong radial mixture gradient input are measured by regulating the fuel-air ratio, while the influence of the variation of the imposed swirl is studied for constant fuel injections. Large Eddy Simulations are performed with the Fluent software using an in-house modified EDC combustion submodel and a developed 10-step propane/NOx oxidation mechanism.

Large Scale Computations of reacting flows with reduced multi-step chemistry and higher order models.

Hydrocarbons combustion is an important phenomenon in energy production. In Internal Combustion Systems, combustion usually takes place within and strongly interacts with a turbulent flow and the adequate description of the process requires consideration of large number of fluid and chemical parameters. Direct or Semi-Direct Numerical Simulations (e.g. DNS, LES) of turbulent reacting flows offer a promising tool toward the understanding of these complex flows. The full potential of these techniques can best be realized when sufficiently complex and realistic but flexible combustion, chemistry and transport models are exploited. Turbulent combustion submodels and tractable chemical schemes for the oxidation of CH$_4$ (natural gas), C$_3$H$_8$ (LPG) or higher hydrocarbons or mixtures (Diesel, surrogates etc) including NO$_X$ and soot production mechanisms are therefore derived and tested in laminar and turbulent protype flows and then applied to practical industrial configurations such as knock prevention and end gas ignition in Internal Combustion Engines.

Example - Reduced C$_3$H$_8$ mechanism for use in IC Engines

C$_3$H$_8$ +O$_2$ +H$_2$O +H$_2$ $\rightarrow$ 3CO + 6 H$_2$,
CO + H$_2$O $\leftrightarrow$ CO$_2$ + H$_2$,
3H$_2$ +O$_2$ $\leftrightarrow$ 2H$_2$O + 2 H,$
2H + M $\rightarrow$ H$_2$,
C$_3$H$_8$ +3H$_2$O +4H $\rightarrow$ 3CO +9H$_2$,
0C$_3$H$_8$ +2CO +H$_2$ $\leftrightarrow$ C$_2$H$_2$ +O$_2$,
0.5 N$_2$ +2O $\leftrightarrow$ NO + O,
2O + M $\rightarrow$ O$_2$,
2O +0H$_2$O $\leftrightarrow$ O$_2$ +0H$_2$,
0C$_3$H$_8$ +N$_2$ +O$_2$ +0H$_2$O $\leftrightarrow$ NO $\rightarrow$ 0C$_3$H$_8$ +2NO
Laminar and turbulent multistep chemistry
Large Scale Simulations of laboratory flames

Internal Combustion Engine Computations,
Diesel engine in-cylinder simulations using a mesh with pre-chamber and swirl.

Typical mesh arrangements in industrial reacting flow computations.