Politecnico di Torino

Finanziato dall'Unione europea NextGenerationEU | Ministero dell'Università e della Ricerca | Italiadomani PIANO NAZIONALE DI RIPRESA E RESILIENZA

ENERGETICS

MUR DM 117/Punch - NUMERICAL INVESTIGATION OF H2-AIR MIXTURE FORMATION IN INNOVATIVE GREEN-HYDROGEN DIRECT-INJECTION ENGINES

Funded By	POLITECNICO DI TORINO - AMMINISTRAZIONE CENTRALE [P.iva/CF:00518460019] MINISTERO DELL'UNIVERSITA' E DELLA RICERCA [P.iva/CF:97429780584] PUNCH TORINO S.P.A. [P.iva/CF:09075820010]

Supervisor	FERRARI ALESSANDRO - alessandro.ferrari@polito.it

Contact	FERRARI ALESSANDRO - alessandro.ferrari@polito.it Alberto Vassallo - PUNCH TORINO S.P.A.

Context of the research activity	The lean combustion direct injection (DI) H2 engine is considered an optimal solution to eliminate the risk of irregular combustion, such as backfiring and pre-ignition, as well as to improve thermal efficiency, volumetric efficiency and NOx engine out emissions. Numerical studies on DI-H2 mixture formation are not still spread, mainly due to the complexity of modeling the mixing characteristics of the under-expanded jets. The research aims at developing an efficient and accurate 3D numerical model of the H2-air mixture formation in DI engines. Progetto finanziato nell'ambito del PNRR – DM 117/2023 - CUP E14D23001950004

	A 3D numerical model will be developed with CONVERGE or other equivalent code to analyze the mixture formation in a DI spark ignited engine fueled with gaseous hydrogen. The considered injector is a GDI sample mounted in side location on the engine head and the injector tip can be installed with a possible recess with respect to the combustion chamber walls. The main objectives of the research activity are the following ones: • validation of the developed numerical 3D model with available experimental data (or images) on H2 jet for different engine working conditions; • numerical analysis of the H2 direct injection for medium (50 bar) and high (100 bar) injection pressure levels. The time histories of penetration length, jet cone angle and jet area will be calculated under different engine operating

Objectives	conditions (expressed in terms of injection pressure, energizing time, start of injection, swirl number, engine speed, etc.); • accurate simulation of the under-expanded flows (these can include Prandtl-Meyer rarefaction fans, oblique shocks and possible Mach disk) and of their effects on mixture formation. The under-expanded region occurs when the ratio between the injection pressure and the in-cylinder pressure is higher than the critical value (this is equal to 1.889 for hydrogen). There are two major flow categories of under-expanded jets: “moderately under-expanded” jets, which are characterized by the “diamond configuration”, and “highly under-expanded jets”, in which a Mach disk is present. For a circular section of the nozzle, the transition from moderately under-expanded to highly under-expanded jet occurs for a ratio of the injection pressure to the in-cylinder pressure ratio between 2 and 3; • optimization of the nozzle layout, the injector position and its orientation. With regard to the nozzle configuration, parametrical analyses will be carried out with the 3D model on the orifice diameter (for single hole injectors), on the aspect ratio of the injection hole, on the number of injection holes (for multi-hole nozzles), on the injection angle (angle of the injection hole axes with respect to the injector axis) and on other features of interest for Punch Torino SpA. Furthermore, both outward- and inward-opening nozzles will be considered. As far as the injector position and its orientation is concerned, parametrical tests will be performed with the 3D model on the injector angle (angle between injector and cylinder axes), on the injector tip recess or protrusion and on other design features of interest for Punch Torino SpA. The experimental mass flow-rate and in-cylinder pressure time histories will be used as boundary conditions for the 3D model. The simulation should cover two cycles: the first cycle starts before intake valve opening and occurs without injection, in order to develop the flow inside the cylinder and be able to replicate the experimental conditions at the start of injection of the next cycle. The second cycle includes the injection process, which starts after intake valve closure, and ends at TDC since combustion is not simulated. The Reynolds-averaged Navier–Stokes (RANS) technique is recommended to achieve an initial understanding of the mixing process in H2-fuelled engines since it has been shown to still represent the best compromise between simulation time and result quality, compared to the Large-Eddy Simulation (LES) method. The proposed RANS techniques for the 3D model include, standard k-e, realizable k-e, renormalization group (RNG) k-e, and Reynolds Stress Model (RSM), which are all available in CONVERGE. The penetration length and the cone jet angle obtained by employing different turbulence models will be compared with available experimental data on the H2 jet. High grid-resolution will be required to characterize properly the flow structures. In fact, very fine meshes and low time steps must be adopted to capture the features of the under-expanded region: the supersonic flow creates an oblique shock structure of barrel-shape, which can include a disk-shaped vertical shock (the so-called Mach disk), and reflected shocks. Furthermore, a multi-hole nozzle can create an intense jet-to-jet interaction, with all jets merging into a single jet immediately downstream of the under-expanded region. This phenomenon (Coanda effect) is even more challenging for numerical simulation and requires an extra high level of detail in numerical simulation and grid resolution, especially regarding the fields near the injector nozzle exit (adaptive mesh refinement will be adopted).

Skills and competencies	The candidate should know the fundamentals of thermodynamics, fluid mechanics and internal combustion engines. Furthermore, he should have an adequate knowledge of computational fluid dynamics, with particular

for the development of the activity	reference to the numerical treatment of under-expanded flows and shocks. The experience of the candidate in the realization of home-made thermo-fluid dynamics codes, applying state-of-the-art numerical techniques for the prediction of shocks or similar phenomena, will be evaluated.