Evaluating and reducing the risk of hydrogen leakage in an aircraft cabin using CFD analysis

by Marco Corti[1], Leonardo Lecce[2]
1. EnginSoft - 2. Novotech

Introduction

The transition to more sustainable aviation is driving the industry to explore new technological frontiers, with hydrogen emerging as one of the most promising candidates for future propulsion. Hydrogen fuel cells represent a potentially revolutionary solution that combines energy efficiency with zero emissions. However, implementing this technology in a specialized sector such as aviation that has multiple risk factors raises important safety issues which require in-depth analysis to prevent and mitigate accidents. Several projects have been developed and are underway to implement fuel cells in the aviation sector in aircraft ranging from small two-seaters to large commercial aircraft.

SERENA, an abbreviation derived from the Italian name of the project (Sviluppo di architetture propulsive ad Emissioni zeRo per l'aviazione gENerAle), thanks to the joint efforts of Distretto Tecnologico Aerospaziale (DTA), EnginSoft, Novotech and Università del Salento – Department of Innovation Engineering, focuses on the development of zero-emission propulsion architectures for general aviation and aims to develop an all-electric propulsion system using a combination of batteries and fuel cells for the Seagull, a VDS (recreational or sport flight) category aircraft manufactured by Novotech. The aircraft stores hydrogen gas in a 350-bar cylinder located in the fuselage behind the pilot seats. The use of pressurized gas, combined with the fact that hydrogen has a low activation energy and a wide range of concentrations (from 4% to 75%) to form an explosive mixture with air, requires careful risk assessment and mitigation. Databases such as HIAD 2.1 provide an overview of the accidents caused by hydrogen usage in different industries which offers a statistical sample and reference for risk reduction.

When hydrogen tanks are used, the malfunctioning or rupture of the gas supply valve is a frequent cause of accidents. With this in mind, the CFD (computational fluid dynamics) activity within the SERENA project focused on simulating a valve rupture scenario resulting in the release of hydrogen into the aircraft cabin during flight.

The aim of the CFD modelling was to assess whether the average hydrogen mole fraction in the cabin would remain below 1% and whether the combustion danger zone (identified as the area with a hydrogen mole fraction greater than 4%) would remain confined to a small area near the leak.

Design requirements and reference standards

Starting from the definition of a series of typical missions (take-off, climb, cruise, descent) of the Seagull aircraft and the data available on its aerodynamic and propeller characteristics, a specific "Mission Performance Calculator" procedure was developed. The power requirements, the total energy required to carry out the various missions, the autonomies, and the mission times were calculated by modifying the main parameters (speed, pitch, cruising altitude, and duration) that uniquely identify each mission. Using this procedure, twelve different typical missions varying in duration (between 30' and 90'), cruising speed, flight altitude, etc. were analysed, after which the power requirements for each flight phase, and for the mission as a whole, were derived.

Project partner UniSalento used this data to define the propulsion system based on an electric motor powered by a suitable fuel cell and a corresponding hydrogen gas tank. It also addressed the certification and safety issues relating to the use of hydrogen to power the fuel cell. In this regard and due to the absence of legislation applicable to light aircraft, UniSalento examined the available regulations (ISO/TC 197, EC ATEX and EU TPED directives) governing the use of hydrogen in other sectors, as well as recent FAA and EASA reports on the use of fuel cells and hydrogen tanks.

Modelling of the cabin housing the propulsion system

A preliminary analysis was performed assuming a cylindrical tank and the presence of two openings at the top and side of the aircraft to provide natural ventilation. Bodies in the vicinity of the tank were also considered as they significantly influence the movement of the fluid. The exit velocity of the leak was estimated using FLOWNEX software for a tank with a known pressure of 350 bar, assuming a cabin pressure of 1atm and a leak diameter of 2.5mm. The exit velocity of the tank leak is u=1,103m/s. The analysis, performed using Ansys Fluent, has the following physical modelling characteristics:

- **Transient**: the analysis evaluates the temporal evolution of the gases within the domain. At the initial moment, a volume consisting only of air is considered, while at later moments the cabin begins to contain hydrogen due to the outflow from the tank.
- **Multicomponent**: a multicomponent simulation, i.e. which evaluates the model as a mixture of fluids, is necessary to evaluate the hydrogen concentration. In the case of the actual simulations, the mixture considered consists of air and hydrogen. The mole fractions X_{H2} and X_{air} are the quantities considered to evaluate the composition of the mixture in the different regions of space
- **Gravitational**: air and hydrogen have quite different densities, so the gravitational field must be considered as it has a significant effect on the dynamics of the gases inside the cabin.
- **Turbulent**: the jet of the leak exiting the tank has a very high velocity due to the considerable difference between the pressure in the cylinder and the pressure in the cabin, resulting in turbulent flow. The difference in density between air and hydrogen is another factor that causes turbulent flow even in regions far from the leak, so appropriate modelling was introduced to account for this physical aspect. Fluent has a variety of models for turbulence. We used the k-ω SST model for this simulation. This method of turbulence resolution belongs to the family of URANS (unsteady Reynolds averaged Navier Stokes) models and is optimal among URANS models because it provides accurate values for many flow regimes. It is the most robust URANS turbulence model because it combines the k-ε and the k-ω standards: the former gives good results at a distance from solid walls (e.g. free jets) but loses accuracy near objects, whereas the k-ω standard provides a hybrid solution between k-ω and k-ε weighted by the distance from the wall. The k-ω prediction outweighs k-ε at proximity, and vice versa at distance. The merits of the two models are thus combined into one.
- **Isothermal**: heat exchange with any heat sources inside the aircraft or with the external environment is assumed to be negligible. The temperature is therefore constant at 15°C, while the surfaces defining the volume are adiabatic.

We used Fluent meshing to discretize the geometry. We paid special attention to mesh densification to accurately represent the complex hull surfaces and the very small size of the leak. We used a poly-hexcore mesh with multiple layers of prisms to accurately capture the velocity gradients present near the walls. The polyhedra provide a transition layer between the prisms and hexahedra.

The hexahedra fill the rest of the volume and are the main cell type in the analysis. We filled the volume with hexahedra instead of polyhedra and, simultaneously, we thickened the computational grid area in front of the fuel tank leak in order to minimize the numerical diffusion error and to optimize the accuracy of the calculations. A sensitivity analysis of the results with respect to the number of elements was conducted using different mesh sizes for the computational grid. A mesh of 4,634,968 cells provided values that were independent of the size of the computational grid and was therefore chosen to illustrate the results of the analysis.

CFD simulation of hydrogen dispersion

The results of the analysis show that the hydrogen jet emerging from the leak is limited in extent by the object situated in front of it. As the jet hits the surface of this object it shows an abrupt change in flow direction and a loss of momentum. As a result of the sharp reduction in the jet's inertia, buoyancy forces dominate the flow dynamics and tend to push the hydrogen into the upper region of the hull. Buoyancy also causes a recirculation zone below the tank. The speed of the hydrogen jet exiting the leak reduces sharply from a velocity of 1,102m/s to a velocity of 100m/s in a distance of 40cm. The hydrogen escaping the tank sets the surrounding air in motion, causing it to expand and simultaneously reduces the velocity of the jet. The reduction is very

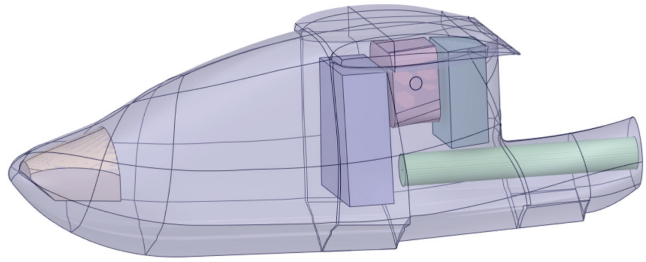

Fig.1. The Seagull's geometry with natural ventilation used for analysis.

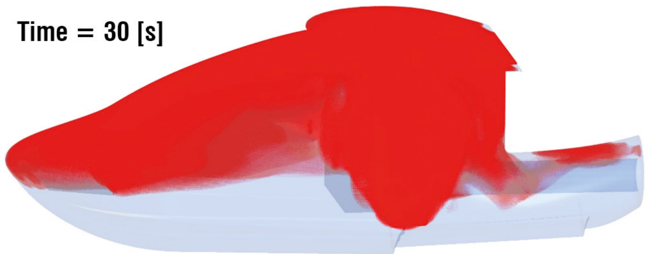

Fig.2. Combustion risk at instant t = 30, region s (natural ventilation case).

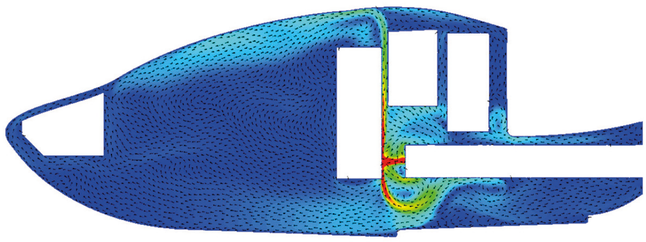

Fig.3. Velocity field in the plane of symmetry (scale limited to 2m/s) for the case with natural ventilation.

abrupt up to a distance of 10cm from the leak and then follows a linear law. The velocity field does not vary significantly over time but remains practically the same at different times, especially in the region of higher velocities, where the motion is caused by the jet.

Analysing the average mole fraction of hydrogen, we see that the concentration of the hydrogen gas increases significantly with time according to a linear law. In just over 5 seconds, it exceeds the target value of an average mole fraction of less than 1%. After 30 seconds, 4.8% of the gas in the cabin is hydrogen. At 2s the concentration is closely tied to the jet dynamics. In the subsequent instants, the hydrogen tends to occupy the upper region of the domain and then moves towards the front region of the fuselage. In the instants that follow, a progressive stratification is observed, with hydrogen also significantly occupying the lower region of the cabin. The results show that after 30s, approximately 70% of the cabin is at risk of combustion.

Risk analysis and mitigation

The results showed that the natural ventilation could not cope with the high flow of hydrogen into the cabin, leading to unacceptable conditions after a few seconds. It was therefore necessary to modify the scenario by considering forced ventilation.

In the new configuration, one of the surfaces that allowed the mixture to exit the domain was modelled as an inlet and the flow rate was set to simulate forced ventilation. Geometrical changes were also made to the initial analysis considering the tank provided by Novotech, modifying the layout of the internal bodies, and introducing baffles in the cabin. The new analysis was carried out in several steps:

- The variation in results with or without the baffles inside the aircraft was evaluated at a ventilation flow rate of 1,500m^3/h. Two different diameters for the ventilation inlet and outlet (160mm and 240mm) were tested for the different cases, as well as two different placements: a "front" placement between the two baffles and a "rear" placement behind both baffles. The diameters of the vents considered are larger than those previously defined for the natural ventilation to reduce the flow velocity in and out of the vents. The forced ventilation significantly improves the results compared to natural ventilation by allowing the average

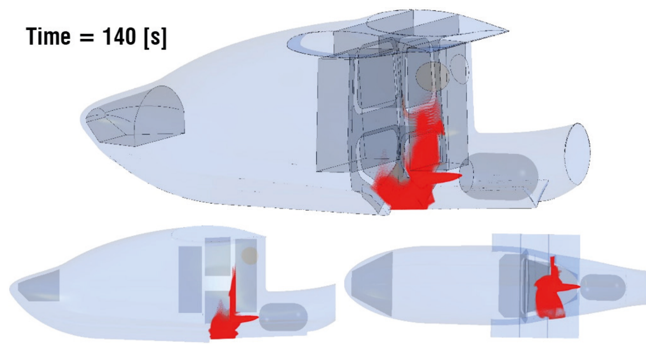

Fig.4. Combustion risk region after 140s with forced ventilation (Case D=240mm, and rear position).

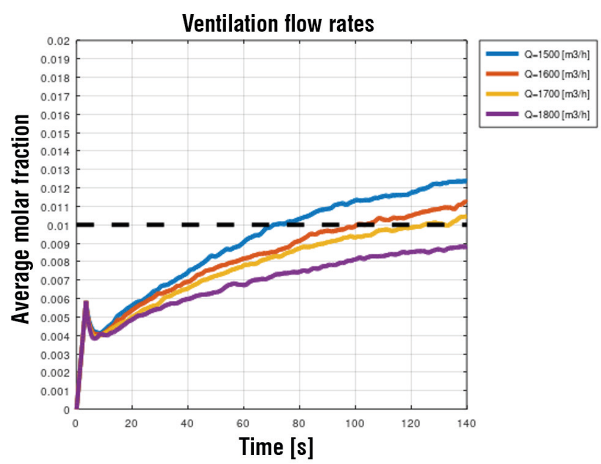

Fig.5. Average hydrogen concentration in the cabin at different forced ventilation flow rates.

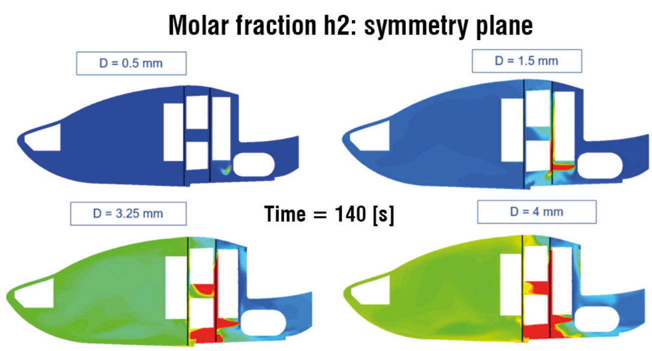

Fig.6. Hydrogen mole fraction for different leak sizes (forced ventilation).

hydrogen concentration to tend towards an asymptote after the initial increase. The area at risk of combustion also remains confined in a time-constrained space. Comparing the various cases, the baffles have a beneficial effect on the amount of flammable gas dispersed in the cabin. The best case achieved is a setup with baffles and a ventilation inlet/outlet diameter of 240mm in a rear placement position. This setup exceeds the target limit of 1% later than the other cases (79s) and remains below 1.2% at 140s. This geometric configuration will be considered for further work.

- The analysis performed in the previous step assumed that the ventilation was already active at the time of tank rupture. We then wanted to introduce a model into the analysis in which the ventilation only activated after a pair of sensors detected the hydrogen leak. The sensors were modelled as point detectors positioned close to the fuselage walls and mirrored with respect to the aircraft's plane of symmetry. Four different sensor placements were considered. Ventilation was activated at time instant t_v, given by the sum of the sensor leak detection time t_s and a system response delay Δt of 3s. The best position for the sensors was "B", which allows the ventilation to be activated after only 3.5s, thus preventing the concentration from exceeding 1% before ventilation intervenes. The results show that positioning the sensors in front of both baffles, as in case D, should be avoided. In this scenario, ventilation activation does not take place until one minute after the situation in the zone has deteriorated (the average hydrogen content is higher than 9%). Sensor position B is used for the rest of the activity.
- Once the geometry and position of the sensors had been determined, the minimum ventilation flow rate was assessed so that the average hydrogen concentration would always remain below 1%. A ventilation system with a flow rate of 1,800m^3/h meets this criterion by keeping the value below 1%, even after a 30-minute time window.

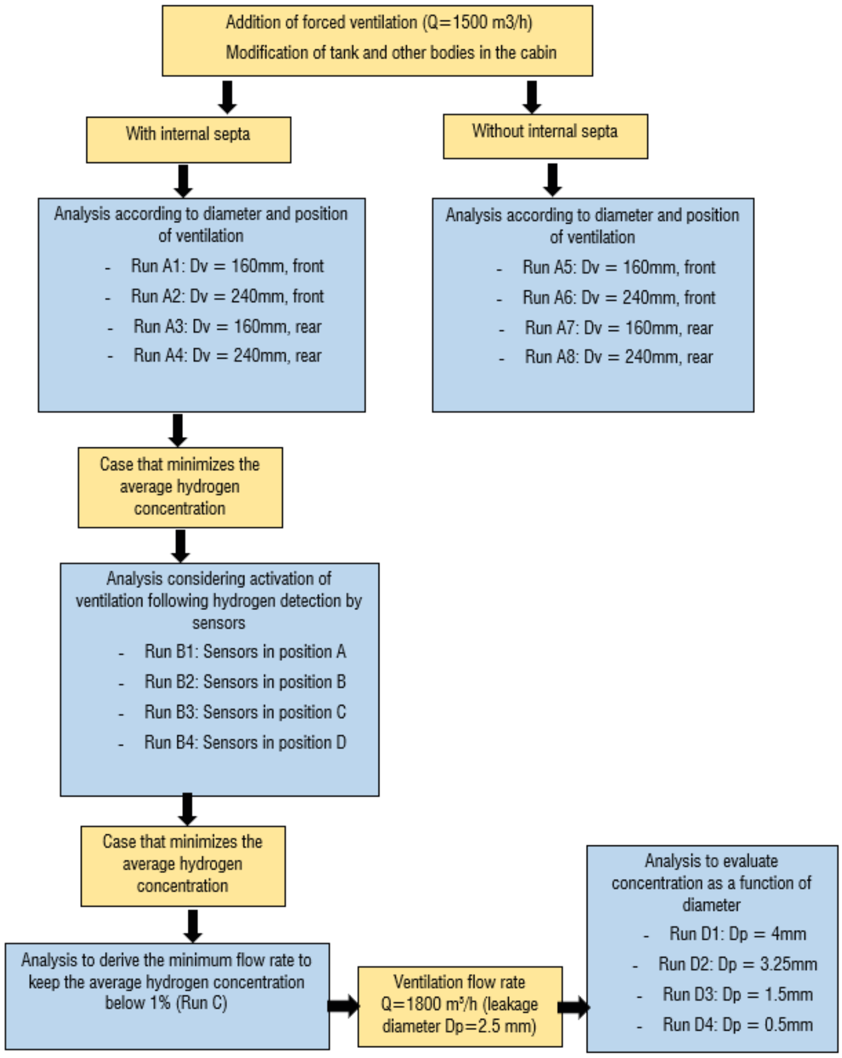

Fig.7. Logical scheme followed to carry out the simulations.

- After defining this new value for the ventilation flow rate, a sensitivity analysis was carried out by evaluating different hydrogen leak sizes, both larger and smaller than the nominal leak used (2.5mm)

A mesh of 4,634,968 cells provided values that were independent of the size of the computational grid and was therefore chosen to illustrate the results of the analysis.

Conclusion

The CFD analyses conducted assessed the risk of hydrogen leakage inside the Seagull cabin following the rupture of a gas fuel tank valve. The analysis of the case with natural ventilation showed that a large area was at risk of combustion after a few seconds. By switching to forced ventilation and following a strategy of optimizing ventilation geometry and sensor response times, it was possible to maintain a hydrogen concentration of less than 1% at a flow rate of 1,800m3/h. Lastly, a sensitivity analysis was carried out to compare the results for leaks with larger and smaller diameters.

Acknowledgement

This work was funded by the European Union - NextGenerationEU - Italian National Recovery and Resilience Plan (NRRP) - M2C2.3.5.

Finanziato dall'Unione europea
NextGenerationEU

MINISTERO DELL'AMBIENTE E DELLA SICUREZZA ENERGETICA

For more information:
Vito Primavera – EnginSoft
v.primavera@enginsoft.com