CFD analysis of aircraft fuel tanks thermal behaviour

C Zilio¹, G A Longo¹, G Pernigotto², F Chiacchio³, P Borrelli³ and E D’Errico³

¹ University of Padova, Dept. of Management and Engineering, S.Ila S. Nicola, 3, I-36100 Vicenza, Italy
² Free University of Bozen-Bolzano, Faculty of Science and Technology, Piazza Università, 5 I-39100 Bolzano, Italy
³ Leonardo, Aircraft Division, Viale dell’Aeronautica, Pomigliano d’Arco, Italy

E-mail: tony@gest.unipd.it

Abstract. This work is carried out within the FP7 European research project TOICA (Thermal Overall Integrated Conception of Aircraft, http://www.toica-fp7.eu/). One of the tasks foreseen for the TOICA project is the analysis of fuel tanks as possible heat sinks for future aircrafts. In particular, in the present paper, commercial regional aircraft is considered as case study and CFD analysis with the commercial code STAR-CCM+ is performed in order to identify the potential capability to use fuel stored in the tanks as a heat sink for waste heat dissipated by other systems. The complex physical phenomena that characterize the heat transfer inside liquid fuel, at the fuel-ullage interface and inside the ullage are outlined. Boundary conditions, including the effect of different ground and flight conditions, are implemented in the numerical simulation approach. The analysis is implemented for a portion of aluminium wing fuel tank, including the leading edge effects. Effect of liquid fuel transfer among different tank compartments and the air flow in the ullage is included. According to Fuel Tank Flammability Assessment Method (FTFAM) proposed by the Federal Aviation Administration, the results are exploited in terms of exponential time constants and fuel temperature difference to the ambient for the different cases investigated.

1. Introduction

In the recent years aeronautical industry has concentrated its design challenges towards a “More Electric Aircraft” concept that implies the reduction of the traditional hydraulic and pneumatic sub-systems and the introduction of electrically driven devices.

As a consequence, aircraft electrical consumption has increased by a factor of five and new engines, whilst more efficient, now produce much more heat. Consequently, a proper thermal management on new aircrafts has become a real challenge. By managing and understanding thermal behaviour early in any aircraft design process, some of the issues related to heat flows towards aircraft components and systems can be abated [1].

Within FP7 research framework, EC launched the Project TOICA (Thermal Overall Conception of Aircraft) with the main target of developing new concepts for improved thermal load management for aircraft components, systems or equipment, which will integrate innovative cooling technologies and products. In particular, TOICA contributed to:

• Improve the overall multidisciplinary conception of aircraft during the architecture phases
• Optimize the overall energy management of the aircraft through a reduction of the aircraft energy consumption
• Reduce thermal constraints on systems and structure, and thermal integrated risks
• Reduce weight and complexity through a fully integrated structure and thermal design of systems.
Among the different “Use cases” analyzed by the TOICAm consortium, different approaches and methods were investigated to model the fuel tank thermal behaviour. The idea of transferring waste heat to the fuel in the aircraft tanks is relatively old and some pioneering works can be retrieved in the literature. For example, in [2] it was already clearly observed that fuel thermal modeling is a key enabler to assess the capability of the system architecture to use fuel as a heat sink. In that report, a simplified analytical model of the thermal behavior of aviation fuel inside a tank was proposed basing on tests run with a mock-up fuel tank.

Several other analytical models have been proposed in the last decades, thanks to the availability of more sophisticated simulation platforms and the more and more increasing calculation capabilities of computers. However, these methods can be considered suitable only for early-phase trade studies to determine the feasibility of an aircraft [3].

When a more detailed modeling approach is needed, it is possible to consider computational fluid dynamics (CFD) models. Recently, Qian and co-authors [4] coupled the numerical results of FLUENT Volume Of Fluid (VOF) analysis on thermal behavior of jet fuel in the tank with a mono-dimensional dynamic simulation tool (FLOWMASTER). FLOWMASTER was used to build a mono-dimensional simulation of aircraft integrated thermal management, including the convective heat transfer coefficients calculated by using FLUENT. The authors concluded that it is possible to realize thermal simulation of aircraft integrated thermal management without flight tests, in order to provide guidance to the design.

To comply with new civil certification regulation [5], the aircraft fleet fuel tank flammability exposure level must be evaluated. The flammability assessment can be performed using the Federal Aviation Administration (FAA) Fuel Tank Flammability Assessment Method (FTFAM). This method uses Monte Carlo statistical approach to generate flammability data for certain unknown variables over known distributions for a large number of flights. The parameters required to run the flammability assessment can be grouped in the following categories: fuel tank surrounding environment, mission data, fuel properties, and the thermal characteristics of the fuel tank. Once these four components are determined, whether by user input or by Monte Carlo calculations, the model can then determine whether the ullage of the fuel tank for each time increment of flight is flammable or not; and furthermore, the percent of the mission time that the fuel tank is flammable. The flight data, required to perform the Monte Carlo calculations, are: the cruise Mach number, altitude steps and tank ram recovery. The airplane data, required to perform the Monte Carlo calculations, includes: the maximum range of the aircraft, the number of engines, and the Outside Air Temperature (OAT) cutoff limit. Other data that must be fixed by the designer regards fuel tank usage information like the tank full and empty times and the engine start times. Finally, the fuel tank thermal data, required to perform the Monte Carlo calculations, are the temperature differential to both ambient and Total Air Temperature (TAT) as well as a number of exponential time constants for the six cases shown in the Table 1. According to FAA [5], the fuel is charged in the tank at a given uniform temperature. Then the whole tank (including liquid fuel, ullage and solid structure) undergoes the thermal solicitation of the surrounding environment and of any possible internal heat source. Through the analysis of the transient response of the system it is possible to estimate the exponential time constant of the fuel considering different levels of the fuel inside the tank (typically, near full and near empty) with reference to both ground and flight conditions. In addition, for ground conditions, these values must be known with and without the engines running. Hence, there are six exponential time constants required as input, which define how the fuel in the tank heats or cools in response to heat input.

Using experimental values of the exponential time constants is usually unfeasible in the early phase of aircraft design, hence analytical or numerical methods should be employed for estimating the six exponential constants.

The heat and mass transfer processes inside the fuel and the ullage in a wing tank is rather complex. Figure 1, modified from [2], is an attempt to highlight the principal mechanisms.
Table 1. FAA Cases for the fuel tank flammability estimation.

| Case | Fuel Quantity         |
|------|-----------------------|
| 1    | Ground-Engine OFF     | Near empty¹       |
| 2    | Ground-Engine OFF     | Near full         |
| 3    | Ground-Engine ON      | Near empty¹       |
| 4    | Ground-Engine ON      | Near full         |
| 5    | Cruise-Engine ON      | Near empty¹       |
| 6    | Cruise-Engine ON      | Near full         |

¹ Near empty does not refer to the portion of tank, the so called “collector tank”, where fuel pump is installed. This part is always full, in normal operation with the engine running.

Figure 1. Main heat and mass transfer mechanisms of the fuel in the tank.

Given the rather complex mechanisms inside the tank and the complex interactions of the tank with surrounding components in the wing and with the external ambient, a simple model can hardly properly capture local phenomena that can lead to hazardous conditions in terms of flammability. CFD is an option to evaluate the time constants requested by FAA regulations and at the same time to properly map local temperatures inside the tank.
In section 2 of this paper, the modeling approach of a portion of the fuel tank of a commercial aircraft is outlined. Furthermore, the relevant boundary conditions for both “ground” and flight” operations are defined according to typical airborne working conditions.

In section 3, the main features of the CFD numerical approach implemented with the software STAR-CCM+ are described in details. An example of the temporal temperature profiles of liquid fuel, ullage and tank structure is reported, in order to exploit the potentiality of the proposed approach to be used in FAA Fuel Tank Flammability Method.

2. Modeling approach and boundary conditions definition

In the present work, reference was made to a regional aircraft. It should be noted that in a regional aircraft, the amount of fuel mass charged in a regional aircraft is lower in comparison to a long haul aircraft. This point can be considered more critical in terms of possible harmful situations as a consequence of an heat input.

In figure 2, an example of a parametric CAD model of the regional turboprop aircraft wing and the fuel tanks is reported. The studies were performed on a section of fuel tank between three consecutive ribs including the fuel collector tank and the adjacent bay i.e., ribs 4, 5 and 6 in figure 3. The reference tank volume section is reported in figure 4. As it can be seen, the aluminum tank structural parts and the leading edge, including the deicing boot, the hot air bleed pipe and the pipe insulation materials were considered in the simulation.

Jet A fuel was considered and its thermophysical properties were defined as function of temperature and pressure according to [6].

In flight, the leading edge zone (i.e. front area adjacent to the tank) is vented by air. This comes directly from the wing NACA air intake and it is conveyed through the holes of the ribs along the leading edge (figure 5). The air flow rate has been estimated according to existing regional aircraft specifications while the temperature was set equal to -12.2 °C.
The reduction of the fuel quantity, caused by the engine feeding, is simulated with the exit from rib 4 of a fuel flow rate equal to the engine consumption (see Figure 6). This reduction is compensated by the fuel inlet from the other compartments (adjacent to bay 5-to-6) through the stringers cut-outs of the rib 6. This effect impacts the heat thermal balance because the temperature of the incoming fuel (i.e. coming from the adjacent compartments) is different from the temperature of the fuel leaving the sump tank even if both flow rates are equal.
The environmental solicitation to the fuel tank has been expressed through a single fictitious temperature, named the sol-air temperature, which can be adopted in order to explore a variety of forcing conditions, also in transient studies.

The solar radiation incident on each segment of the skins profiles has been estimated starting from horizontal diffuse solar irradiance and direct normal irradiance recorded in typical weather data file (like for example International Weather Files for Energy Calculations - IWEC). The chosen calculation model for solar irradiance on tilted surface is the isotropic solar model, which gives good estimations for clear sky conditions (i.e., for days with maximum solar irradiance). The surface of the ground has a reflectivity of 0.2 and the wing profile is oriented towards the equator (e.g., south-oriented for Milan).

For the top skin, an important role in the surface heat balance is played by the infrared radiation towards the sky dome. The top skin profile was split in several segments and for each segment a proper view factor of the sky dome was introduced. The infrared heat transfer coefficient towards the sky dome has been calculated assuming the mean radiant temperature as an average between the OAT and the fictitious sky temperature, assumed as 13 K (tropical regions) or 11 K (temperate regions) lower than OAT.

As regards the surface heat transfer coefficient by radiation, which considers the heat exchanged by radiation with the surrounding environment, it has been calculated considering a mean radiant temperature equal to OAT and is equal to 5 W/(m² K). In flight conditions, it has been neglected.

In ground conditions, the convective heat transfer coefficient is 20 W/(m² K), which corresponds approximately to the case of wind velocity of 4 m/s. In ground conditions, the surface heat transfer coefficient has been kept constant along the profile.

In flight conditions, the local heat transfer coefficient of the wing skin has been calculated according to [7]. For the remaining parts of the investigated fuel tank i.e., the external remain surfaces (Rib 4, Rib 6, Rear spar) and the bleed pipe in the leading edge, proper surface temperature or local heat transfer coefficients were imposed, basing on typical airborne conditions for both ground and flight operations.

3. Numerical approach

The Jet A thermal behavior in the six FAA conditions reported in Table 1 was studied numerically with the CFD code STAR-CCM+. The volume between Rib 4 and Rib 5 (see figure 3) was considered almost full of Jet A, since this is the normal working condition during standard operation of the aircraft. The liquid fuel level inside the tank part between Rib 5 and Rib 6 can vary, according to the mission profile.

The following Table 2 summarizes the simulation approach used for the structural part (including tank, boot and bleed pipe) and the fluid domain.
Table 2. Main features of the CFD simulation approach.

| Domain                  | Material | Number of cells |
|-------------------------|----------|-----------------|
| Structural parts        |          |                |
| Boot                    | Aluminum | ~185000         |
| Insulation Bleed pipe   | Neoprene | ~25000          |
| Fluid parts             | Kevlar   | ~5000           |
|                         | Jet A and air | ~675000       |

Solid parts were studied by adopting the following models: 3D, implicit unsteady (time-step 0.05 s), solid with segregated solid energy, constant density. The surface-to-surface radiation was enabled with gray surfaces hypothesis and view factors calculation.
Regarding fluid domain, the following approach was used: 3D, implicit unsteady (time-step 0.05 s), Volume of Fluid (VOF); multiphase flow with segregated flow and multiphase temperature (two Eulerian phases: liquid Jet A and air). In order to reduce the complexity of the calculations, no multiphase interactions were considered. K-\( \varepsilon \) turbulence model was imposed and gravity effects were considered.
As mentioned before, relevant Jet A and air properties were considered as a function of local temperature and pressure.
The simulations were run with an High Performance Calculation cluster (HPC) with 180 cores in parallel (Intel Xeon Processor E5620 4C 2.40 GHz 12 MB Cache 1066 MHz 80w).
3.1. Simulation results
Examples of temperature profiles at different times in one longitudinal plane section of the fuel tank are reported in figure 7, while the velocity distribution in the same section after 30 min is reported in figure 8.
In figures 7 and 8, it is possible to appreciate the local temperature values. This feature allows to check the possible subsistence of local hazardous conditions for Jet A fuel flammability. For example, it is evident that the structural stringers inside the tank have a remarkable effect in the heat transfer inside the fuel. The leading edge is considered to have a relevant impact on the fuel tank boundary conditions.

![Temperature distribution in one longitudinal section of the fuel tank, including the leading edge (ground operation).](image-url)
Clearly, the use of the CFD code allows to easily determine the bulk temperature values in order to estimate the exponential time constants as requested by FAA regulation.

In figure 9, the temperature profiles (bulk temperatures of the aluminum structure, of the liquid Jet A fuel in the bay between ribs 4 and 5 and the air temperature in the same bay, of the liquid Jet A fuel between rib 5 to 6 and the air in the same portion) are reported as a function of time with reference to FAA case 5 (i.e. flight conditions, with rib 5 to 6 near empty and rib 4 to 5 almost full of fuel).

In this case, about 300 kg of Jet A fuel are charged in the tank at 15 °C just before take-off. The cruise pressure is 46450 Pa, the bleed pipe temperature is 200 °C, outside air temperature (OAT) is -26.4 °C and total air temperature (TAT) is -12.2 °C. An internal heat source (2 kW) is located inside the fuel in
the bay between rib 4 and 5. This internal heat source is representative of a generic heat load released within the fuel tank.

According to FAA rules [5], time constants should be evaluated for the fuel contained in Rib 4-to-5 bay (R45) and in Rib 5-to-6 (R56). As expected, given the larger amount of fuel mass present in R45 (almost full), the time constant is larger than for R56 (almost empty). The solid orange line represents the liquid Jet A bulk temperature (i.e. considering both bay R56 and R45), as estimated by STAR CCM+. Dotted orange line represents the ullage bulk temperature. This temperature profile is affected also by the air motion coming from the NACA scopes that permit air flow in order to compensate any possible difference between tank inner pressure and ambient pressure.

The aluminum structure bulk temperature never reaches the ambient air temperature (even after 10800 s, the aluminum is about 19 K warmer that the ambient air). This is a consequence of the internal heat load (pump and fuel) and of the heat rejected by the bleed pipe through the leading edge.

It is interesting to observe that fairly constant bulk fuel temperature (solid orange line in figure 9) is achieved only after about 10800 s (i.e. 3 h) that is a period of time usually longer than the duration of a regional aircraft cruise.

Finally, on the basis of the temperature profiles in figure 9, it is possible to calculate the temperature differential with the ambient temperature (OAT) and the total ambient temperature (TAT) as requested by the FAA Fuel Tank Flammability Method.

4. Conclusions
In this paper a comprehensive numerical approach based on STAR-CCM+ CFD software is proposed for the simulation of fuel thermal behavior inside a portion of the tank of a regional aircraft. In the framework of the European project Thermal Overall Integrated Concept of Aircraft (TOICA), several working conditions were investigated in order to simulate both ground and flight situations with the tank almost full or almost empty of fuel. Particular care was given to the definition of suitable and realistic boundary conditions before running the CFD simulations. The considered cruise scenarios were selected according to the Federal Aviation Administration (FAA) rules.

The CFD tool offers the unique capability of mapping the local temperatures in whole domain, thus exploiting potential hazardous regions. The results indicate that the proposed numerical approach can be used for the estimation of fuel time constants and of the fuel temperature difference to the ambient. This set of data can be used as an input for further assessment of fuel tank flammability by means of statistical analysis (e.g. Monte Carlo method).

Furthermore, the proposed numerical approach is a viable option for the implementation of a thermal simulation of aircraft integrated thermal management without flight tests, in order to provide guidance to the design starting from the aircraft architecture definition.

Acknowledgement
The research leading to these results has received funding from the European Union Seventh Framework Programme (FP7/2007-2013) under grant agreement n° 604981.

References
[1] TOICA Website http://www.toica-fp7.eu
[2] Kosvic T C, Zung L B, Gerstein M 1971 Tech. Report AFAPL-TR-71-7
[3] German B J 2012 J. Propulsion Power 28 204 – 10
[4] Su Q, Chang S N, Yang S Y 2016 IEEE/CSAA Int. Conf. Aircraft Utility System October 10-12, 2106 Beijing (China) 774 – 79
[5] Summer S M, Hughes W J 2008 DOT/FAA/AR-05/8
[6] CRC 2014 Aviation Fuel Properties Handbook, CRC Report No. 663
[7] Newton J E, VanFossen G J, Poinsatte P E, DeWitt K J 1988 NASA Tech. Memorandum 100284 AIAA-88-0287