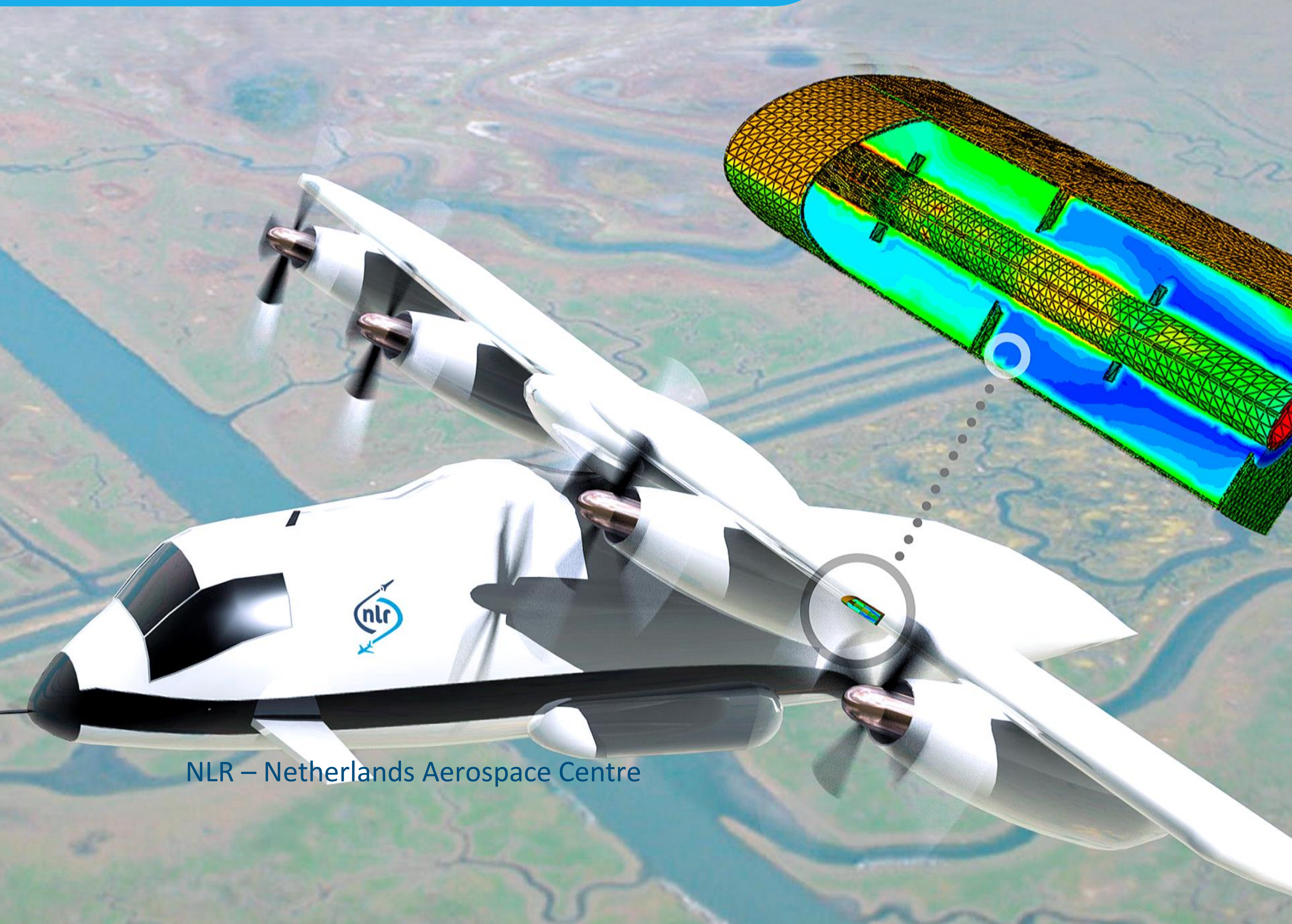
Dedicated to innovation in aerospace

NLR-TP-2016-419 | November 2016

Reducing thermal influence of a bleed pipe near a composite fuel tank wall

CUSTOMER: European Commission



NLR – Netherlands Aerospace Centre

Netherlands Aerospace Centre

NLR is a leading international research centre for aerospace. Bolstered by its multidisciplinary expertise and unrivalled research facilities, NLR provides innovative and integral solutions for the complex challenges in the aerospace sector.

NLR's activities span the full spectrum of Research Development Test & Evaluation (RDT & E). Given NLR's specialist knowledge and facilities, companies turn to NLR for validation, verification, qualification, simulation and evaluation. NLR thereby bridges the gap between research and practical applications, while working for both government and industry at home and abroad.
NLR stands for practical and innovative solutions, technical expertise and a long-term design vision. This allows NLR's cutting edge technology to find its way into successful aerospace programs of OEMs, including Airbus, Embraer and Pilatus. NLR contributes to (military) programs, such as ESA's IXV re-entry vehicle, the F-35, the Apache helicopter, and European programs, including SESAR and Clean Sky 2.
Founded in 1919, and employing some 650 people, NLR achieved a turnover of 73 million euros in 2014, of which three-quarters derived from contract research, and the remaining from government funds.

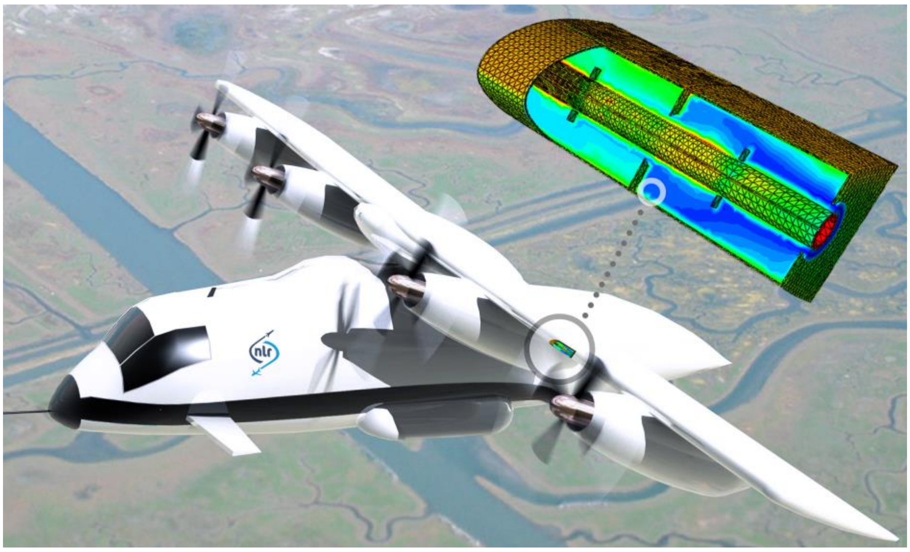
For more information visit: **www.nlr.nl**

EXECUTIVE SUMMARY

UNCLASSIFIED

Dedicated to innovation in aerospace

Reducing thermal influence of a bleed pipe near a composite fuel tank wall



REPORT NUMBER
NLR-TP-2016-419

AUTHOR(S)
S.M. Moghadasi
A.J. de Wit
F. Chiacchio

REPORT CLASSIFICATION
UNCLASSIFIED

DATE
November 2016

KNOWLEDGE AREA(S)
Computational Mechanics and Simulation Technology

DESCRIPTOR(S)
Thermal Model
Multi-Physics
Co-Simulation
Fluid-Structure
Leading Edge

Problem area

Over the years, several fuel tank explosion accidents have highlighted the risks of aircraft fuel tanks. Studies have been carried out by airworthiness authorities that have resulted into additional certification requirements for transport category airplanes with respect to the flammability of the fuel tank. These newly introduced certification requirements necessitate detailed and accurate predictive models for flammability analysis assessment of unconventional fuel tanks. The requirements consider composite materials as unconventional.

Because of the structural and aerodynamic requirements on the aircraft wing design, the enclosures within the wings are rather complex and do not have standard shapes. In particular, the leading edge area is further complicated by the presence of a bleed pipe in the enclosure. The hot bleed air might lead to extreme temperatures in the leading edge and on the fuel tank wall. The prediction of the thermal behaviour in such geometries necessitates 3D methods that are more powerful than the current system analysis methods to model complexities. These detailed 3D models are needed to support the flammability analysis of a fuel tank via predicting the influence of external hot spots near the tank wall.

Within the EU funded TOICA project the thermal behaviour of an entire aircraft is being analysed. Aircraft manufacturers, NLR and their partners in the project intend to change the way thermal studies are performed within the aircraft design and analysis processes. Therefore, parties focus on the creation and management of new aircraft architectures, including thermal analyses and trade-off studies of such architectures.

Description of work

In this study the thermal behaviour of a composite wing leading edge is investigated. A 3D Finite Element model of the structure has been created that takes into account convection, conduction and radiation effects. In addition a 3D Finite Volume model of the air inside the leading edge is created. Via a computational fluid dynamics approach the flow of air and thermal behaviour of the air is modelled. The structure and fluid model are coupled via a co-simulation engine to exchange heat flux and temperature. Different ventilation cases of the leading edge and their impact on the thermal behaviour of the front spar are investigated.

Results and conclusions

Provided that the bleed pipe is located far enough from the spar and covered with decent thermal heat isolation, the composite leading edge structure will not reach extremely high temperatures.

Applicability

The performed study has enabled NLR to perform 3D thermal simulations on leading edge constructed from various materials, such as composites. A flexible and accurate methodology has been developed to support the aerospace industry in thermal behaviour studies.
Furthermore, the models can be used to support a flammability analysis assessment.

GENERAL NOTE
This report contains a paper that is submitted to the EASN Conference 2016, Porto, Portugal, October 18-21, 2016.

NLR
Anthony Fokkerweg 2
1059 CM Amsterdam
p) +31 88 511 3113 f) +31 88 511 3210
e) info@nlr.nl i) www.nlr.nl

Dedicated to innovation in aerospace

NLR-TP-2016-419 | November 2016

Reducing thermal influence of a bleed pipe near a composite fuel tank wall

CUSTOMER: European Commission

AUTHOR(S):

S.M. Moghadasi	Netherlands Aerospace Centre
A.J. de Wit	Netherlands Aerospace Centre
F.Chiacchio	Leonardo

NLR - Netherlands Aerospace Centre

This report contains a paper that is submitted to the EASN Conference 2016, Porto, Portugal, October 18-21, 2016.

CUSTOMER	**European Commission**
CONTRACT NUMBER	**ACP3-GA-2013-604981-TOICA-604981**
OWNER	**NLR + partner(s)**
DIVISION NLR	**Aerospace Vehicles**
DISTRIBUTION	**Unlimited**
CLASSIFICATION OF TITLE	**UNCLASSIFIED**

APPROVED BY :		
AUTHOR	**REVIEWER**	**MANAGING DEPARTMENT**
S.M. Moghadasi	J. Kos	A.A. ten Dam
DATE 2 8 1 1 1 6	DATE 2 8 1 1 1 6	DATE 2 8 1 1 1 6

Contents

This page is intentionally left blank.

REDUCING THERMAL INFLUENCE OF A BLEED PIPE NEAR A COMPOSITE FUEL TANK WALL

S. Marie Moghadasi

NLR - Netherlands Aerospace Centre, Anthony Fokkerweg 2,
1059 CM Amsterdam, The Netherlands
Marie.Moghadasi@nlr.nl
http://www.nlr.org

Albert J. de Wit

NLR - Netherlands Aerospace Centre, Anthony Fokkerweg 2,
1059 CM Amsterdam, The Netherlands
Bert.de.Wit@nlr.nl
http://www.nlr.org

Fabio Chiacchio

Leonardo, Viale dell'Aeronautica,
Pomigliano d'Arco 80038, Italy
Fabio.Chiacchio@leonardocompany.com
http://leonardocompany.com

Abstract

Over the years, several fuel tank explosion accidents have highlighted the risks of aircraft fuel tanks. Studies have been carried out by airworthiness authorities that have resulted into additional certification requirements for transport category airplanes with respect to flammability of the fuel tank. These newly introduced certification requirements necessitate detailed and accurate predictive models for flammability analysis assessment of unconventional fuel tanks (Summer, 2008). The requirements consider composite materials as unconventional.

A fuel tank model consists of an internal analysis of the fuel and fuel vapour and external analysis covering effects of the environment on the tank walls, such as hot spots. A typical hot spot near the wing fuel tank wall is the bleed pipe inside the leading edge. In order to prevent fuel vapours accumulation in case of fuel tank leakage and the tank wall of becoming too hot, the area in front of the tank is ventilated. At present, thermal studies are performed using a computational approach based on SAE AIR 1168-3 standard (International, 1990). This standard consists of a system analysis of aircraft cabin. It was developed to estimate the thermal loads on aircraft cabin and determine the necessary air-conditioning. With a similar approach thermal loads are determined on wing fuel tank and a bulk fuel tank temperature can be calculated. A step towards improving prediction of hot spot thermal effects on the tank wall is via 3D simulation.

In this study the wing leading edge is investigated. Different ventilation cases are outlined and the thermal behaviour of the front spar (tank wall) is investigated. Results of 3D analysis illustrate good insight of the heat transfer and detailed temperature distribution of the spar. Hence, benefits of using 3D analysis are promising. Furthermore, the models can be used to support a flammability analysis assessment.

Keywords Thermal analysis; co-simulation; leading edge ventilation; TOICA.

1. Introduction

Composite materials are poor conductors of heat. The increased usage of such novel materials, in addition to integrating highly heat dissipative and electrical systems in aircraft structures, necessitates studies in the thermal behaviour of aircraft in the early design stages, to avoid building up of heat inside composite aircraft.

In particular, composite fuel tanks and their direct surroundings are of concern. Several fuel tank explosion accidents have highlighted risks in fuel tanks and have led to additional certification requirements for transport category airplanes with respect to flammability of the fuel tank (Summer, 2008). For unconventional fuel tanks, such as composite fuel tank structures, a detailed and accurate predictive model for flammability analysis assessment is required.

Aircraft fuel tanks are usually located inside the wings between the front and rear spar. Furthermore, to increase fuel storage capacity a centre wing tank can be present. The aircraft model that is considered here has a high wing configuration that does not consist of a centre wing tank. In front of the wing fuel tank wall (the front spar) is the wing leading edge. The wing leading edge area is usually designed with aerodynamic, impact damage and environmental conditions in mind.

Because of the structural and aerodynamic requirements on the aircraft wing design, the enclosures within the wings are rather complex and do not have standard shapes. In particular, the leading edge area is further complicated by the presence of a bleed pipe in the enclosure (Moore, Newport, Egan, & Lacarac, 2012). The prediction of the thermal behaviour in such geometries necessitates 3D methods that are more powerful than the current system analysis methods to model complexities.

This paper can be read as follows. In section 2 the design problem is outlined. Section 3 describes the structural and fluid model used for the leading edge. The coupling of both structural and fluid model is described in Section 4. Finally in Section 5 the main results and conclusions are presented.

2. Description of Problem

To determine if thermal problems occur inside the fuel tank because of tank wall heating, a thermal analysis is necessary of the section covering the wing leading edge.

The current design practice is to rely on system model analysis; the thermal condition (temperature profile) is calculated via the computational approach based on the SAE AIR 1168-3 standard (International, SAE AIR 1168/3 Aerodynamic systems engineering and design, 1990). This standard consists of a system analysis of aircraft cabin. It was developed to estimate the thermal loads on aircraft cabin and determine the necessary air-conditioning. With a similar approach thermal loads are determined on wing fuel tank and a bulk fuel tank temperature can be calculated.

In this work we develop 3D models to improve insight into the thermal behaviour of the leading edge and heat transfer into and from the fuel tank. Furthermore, if regions exist in the leading edge that differ significantly from the overall thermal picture of the leading edge these will be visible in a 3D analysis. Finally, the models can be used to support a flammability analysis assessment.

A schematic overview of the wing fuel tank including the leading edge is shown in Fig. 1. An inflow arrow indicates venting of the leading edge.

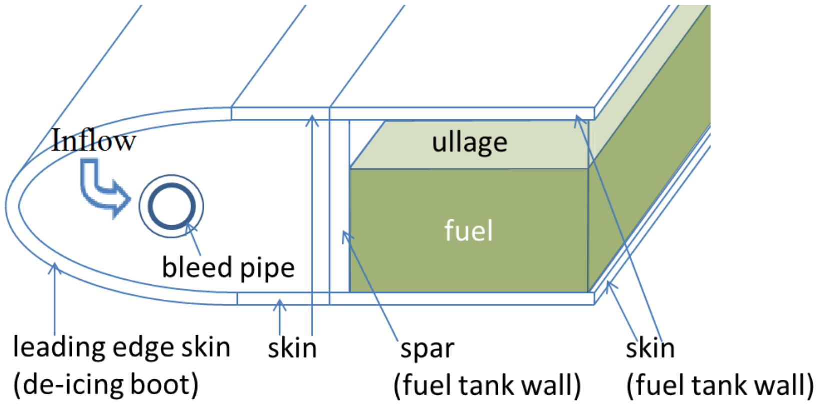


Fig. 1. Cross section of the wing showing wing leading edge, bleed pipe and front part of the fuel tank behind the spar

External heat sources on the fuel tank walls are typically bleed pipes that are running through the wing where the fuel tank is located and solar radiation. Bleed air is hot air that is taken directly from the engine to power on board systems of the aircraft. Because this air is typically 200-250 Celsius the bleed pipe is covered with isolation material. This isolation material reduces the temperature of the external surface of the bleed pipe to temperatures around 100 Celsius. In order to prevent fuel vapours accumulation in case of fuel tank leakage and the tank wall of becoming too hot, the area where the bleed pipe is located is ventilated. The air that is used to ventilate the wing enters and leaves the wing via so called NACA scoops.

A NACA scoop is a common form of low-drag air inlet design located along the wing tip, originally developed by the U.S. National Advisory Committee for Aeronautics (NACA), the precursor to NASA, in 1945. (C.Vale & H.Ringrow, 2011). As an air inlet, the NACA scoop ensures that the differential pressure between the interior of the fuel tanks and the outside atmospheric pressure is kept to a minimum, particularly during climb and descent.

A fluid-structure model is created that captures the effect of heat sources and cooling effects of vents. The bleed pipe located in the wing leading-edge in front of the fuel tank performs as a hot spot. For de-icing purposes, the leading edge is covered with a de-icing boot. Conductivity of the de-icing boot is two orders of magnitude smaller than that of composite material. Hence, the de-icing boot will act as thermal insulation on the leading edge compartment.

3. Thermal Analysis

The structural thermal model uses a Finite Element approach and simulates the structural thermal loading. The fluid thermal model uses a Finite Volume approach and models thermal behaviour of the air inside the leading edge. The two models are coupled to exchange temperature and heat flux values. For the structural surfaces facing the wing's internals the exchange of parameters is accomplished via determining the temperature of the structure first assuming a heat flux coefficient on the internal surfaces in contact with fuel and ullage. Second, the temperature profile of the flow (via CFD) is calculated. Finally, the heat flux profile of the fluid volume is extracted and applied as a boundary condition for the thermal structural analysis (via FEA). These models are simulated in Abaqus standard (Finite Element) and Abaqus (CFD) Computational Fluid Dynamics (Finite Volume), and are coupled through Abaqus co-simulation engine (Abaqus, 2016).

In Fig. 2 an overview is given of the two sections of the wing leading edge structure consisting of skin, spar, and bleed pipe. The fluid model consists of the air inside the wing leading edge bounded by skin, spar and bleed pipe. The hot air inside the bleed pipe is not part of the CFD model but modelled as boundary conditions for the structural FE model. The fuel section is not modelled in this fluid-structure model but taken into account as a boundary condition. Hence, the front spar is the boundary of the structure model. The ullage and fuel temperatures inside the fuel tank have been calculated and provided by University of Padova using a CFD program and are taken into account as boundary conditions.

3.1. *Structural model*

Hot air coming from the engine is running through the bleed pipe. This bleed pipe is heated up and therefore covered with isolation material. The heat that cannot be contained by the isolation material is released via radiation to the surrounding walls and via convection with the air inside the leading edge. Furthermore, heat is released via conduction with the bleed pipe supports that are connected to the front spar wall. In Fig.2 the structure model is illustrated.

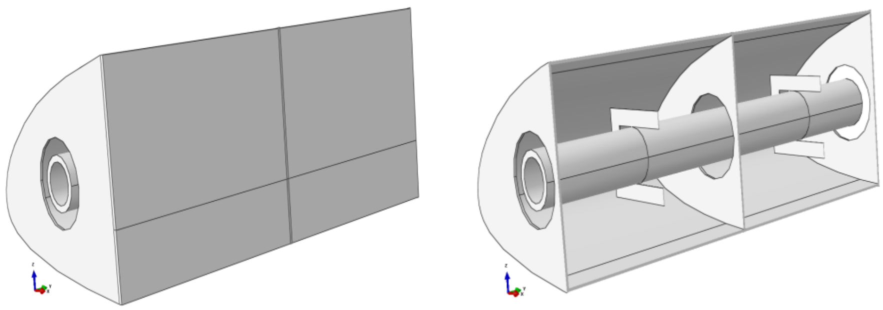


Fig. 2. Structure model of the wing leading edge.

The leading edge model is meshed with linear tetrahedral elements of type DC3D4 (Abaqus, 2016) and the corresponding mesh is shown in Fig. 3.

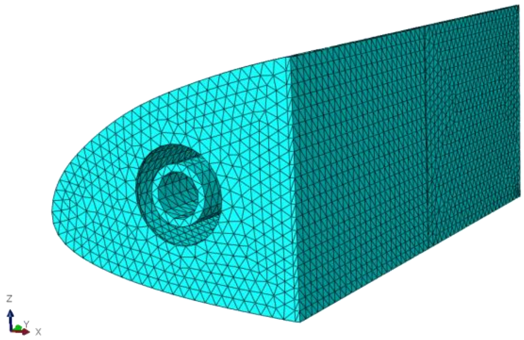


Fig. 3. DC3D4 elements for structural model

Fixed temperatures on the leading edge top and bottom skin (including the de-icing boot) are used. Thermal interaction of the wing front spar with the fuel is modelled via heat transfer coefficients that were obtained from a CFD analysis conducted by the University of Padova for the fuel tank internals, i.e. for ullage and fuel. The interaction with the air inside the leading edge is modelled via a fluid-structure interaction, further described in section 4. Calculated wall temperatures on the Finite Element Analysis (FEA) nodes are transmitted to the fluid calculations and the fluid calculations return heat transfer coefficients on the CFD nodes.

3.2. *Fluid model*

To model the thermal interaction of the wing leading edge structure and the air inside the leading edge, a fluid model is created of the leading edge internals. In Fig.4 the fluid geometry is shown. All the empty space inside the leading edge is covered with the volume mesh.

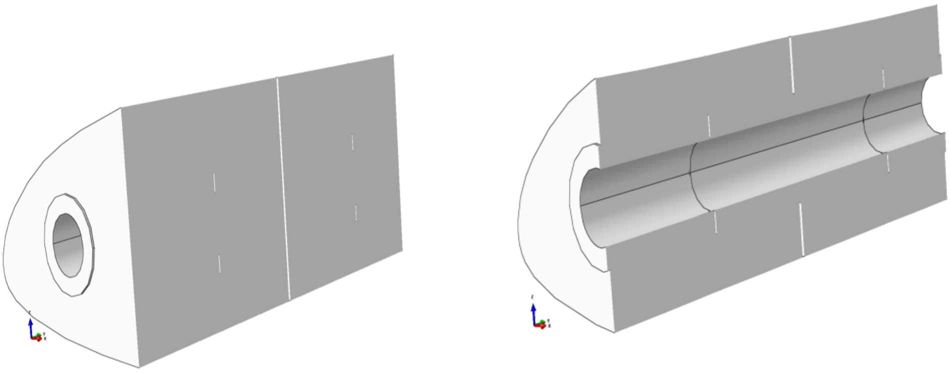


Fig. 4. Fluid volume model of the wing leading edge.

Air inflow comes directly from the wing NACA air intake and it is conveyed through the holes of the ribs along the leading edge. This is shown in Fig. 5. The bleed pipe is smaller than the hole in the rib section. This open area is where the air is entering the wing leading edge in the model. On the opposite side the air is leaving the wing leading edge via the space between the rib and the bleed pipe. The ventilation is taken into account in the model using a constant airflow entering the model on the left. On the right side of the leading edge the rib has an opening and here a constant pressure of 1 bar is applied. The result is a flow of air at constant temperature that cools the leading edge and prevents the leading edge internal from heating up too much by the bleed pipe.

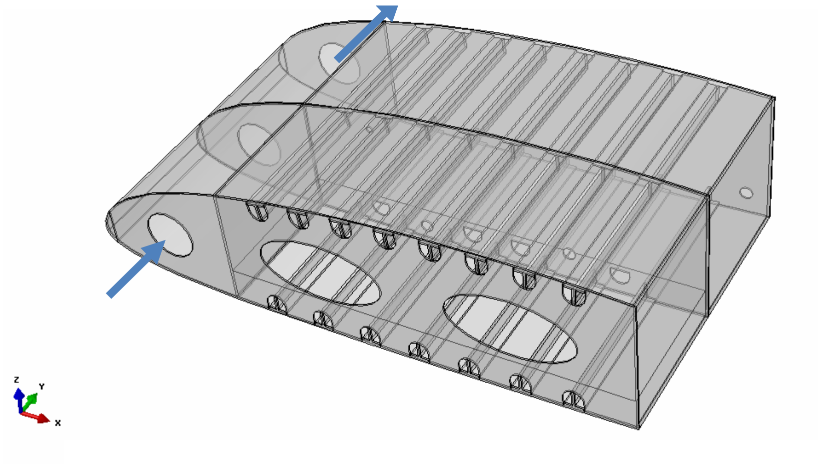


Fig. 5. Wing leading edge NACA air intake.

The fluid model is meshed using fluid linear tetrahedron elements. The corresponding mesh is shown in Fig. 6. The mesh seed is chosen such that it more or less corresponds to the mesh seed that is used for the structure domain.

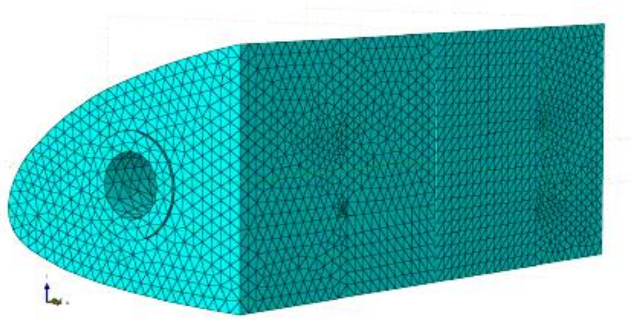


Fig. 6. Mesh of the fluid model.

The Spalart-Allmaras turbulence model is applied that uses wall functions to avoid very fine meshes near the walls. Boundary conditions are no slip conditions where the air touches the structure. Furthermore, the constant fluid flow is prescribed as velocity boundary condition in the opening on left side of the leading edge.

4. Coupled Structural-Fluid Thermal Model for the Leading Edge

A transient analysis is carried out with an initial time step of 1 for the FEA and CFD model. The CFD model uses an automatic time stepping scheme called Courant-Friedrichs-Lewy (CFL) time incrementation method to determine a stable time increment. The time integration method is Galerkin. The numerical procedure to solve the pressure equation is using Bi-Conjugate Gradient stabilized with as a pre-conditioner algebraic multigrid.

The FEA model uses fixed time incrementation, a direct solver and Full Newton. The maximum allowable temperature change per increment is 10 degrees and the maximum allowable emissivity change per increment is 0.1.

The Co-simulation agent uses Gauss-Seidel iterative transient coupling scheme and runs FEA and CFD sequentially (E.L.Blades, E.A.Luke, A.G.Kurkchubashe, & E.M.Collins, 2010). The targeted time step to exchange data for both analyses is the FEA time step. The CFD uses so called sub cycling (multiple time increments to reach a target time step) and the FEA uses a so called lockstep (single time increment to reach a target time step).

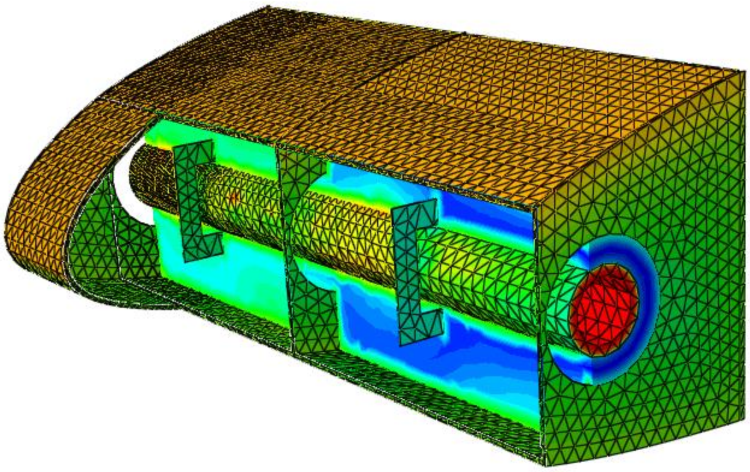


Fig. 7. Cut out of the wing leading edge 3D model

In Fig. 7 a cut-out of the finite element model and a cut-out of the finite volume model are shown to illustrate the resulting thermally coupled fluid-structure model.

4.1. *Simulation conditions*

The heat flow in the leading edge is simulated for a case where the aircraft is on ground with engine on .One fuel tank bay is considered full while the other is half filled with fuel and half ullage, as shown in Fig 8. A fixed outside ambient temperature.is used and other parameters such as leading edge wall temperatures are kept constant while simulations run with different inflow values to study the ventilation effects.

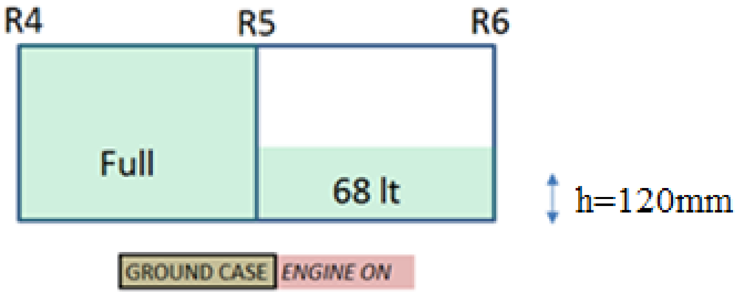


Fig. 8. Schematic overview of the fuel tank levels

The top and bottom skins as well as the wing spar are made of composite material. The curved part of the leading edge consists of a de-icing boot in silicone rubber (neoprene). The bleed pipe is modelled as combination of isolation material and bleed pipe material, and has combined material properties. The material properties of the leading edge are summarised below and illustrated in Fig. 9.

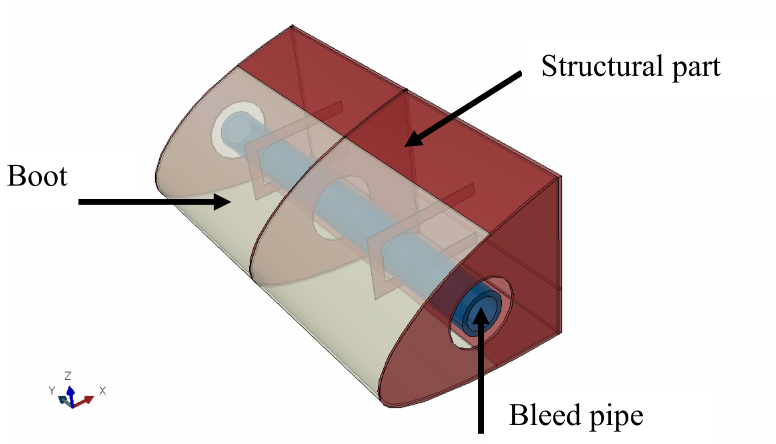


Fig. 9. Leading edge various materials

Structural part:

- Density: 1600 kg m^{-3}
- Thermal conductivity:
  Fibre direction: 6.7 W m^{-1} K^{-1}
  Normal to fibre direction: 0.7 W m^{-1} K^{-1}
- Specific heat: 1400 J kg^{-1} K^{-1}
- Surface emissivity: 0.9

Boot:

- Density: 34 kg m^{-3}
- Thermal conductivity: 0.053 W m^{-1} K^{-1}
- Specific heat: 1200 J kg^{-1} K^{-1}
- Surface emissivity: 0.7

Bleed pipe:

- Density: 16 kg m^{-3}
- Thermal conductivity: 0.035 W m^{-1} K^{-1}
- Specific heat: 800 J kg^{-1} K^{-1}
- Surface emissivity: 0.7

Air properties taken at 293 K:

- Conductivity: 0.0256 W/m-K
- Density: 1.2047 kg/m^3
- Specific Heat: 1006.1 kJ/kg-K
- Kinematic Viscosity: 1.511e^{-5} m^2/s

Cooling of the leading edge is accomplished via a constant mass flow of ambient air into the leading edge. Various combinations of air inflow velocity are chosen as starting condition. The top part of the leading edge is heated up by a constant solar radiation that leads to a 344K temperature of top skin. The bottom part of the leading edge is heated via reflection of solar radiation from the ground leading to a 327K temperature of bottom skin. Hot air (473K) is heating up the bleed pipe which is modelled via sink temperature and convection coefficient. The left tank compartment is full of fuel at a temperature of 330 K while in the right tank compartment the fuel has a temperature of 325 K. In addition, the right tank compartment has a fuel level of 12cm from the tank bottom. The ullage space in the right tank compartment has a temperature of 327K. Different simulations for constant ambient temperature of 293K , 323K and 333K with different inflows are performed. The achieved cooling effects is very small for all these cases and does not exceed 3 degrees.

5. Main Results and Conclusion

With the boundary conditions and initial states as described in previous section a simulation is run for different mass inflows of air. The ambient temperature of the air is 293K for each ventilation case.

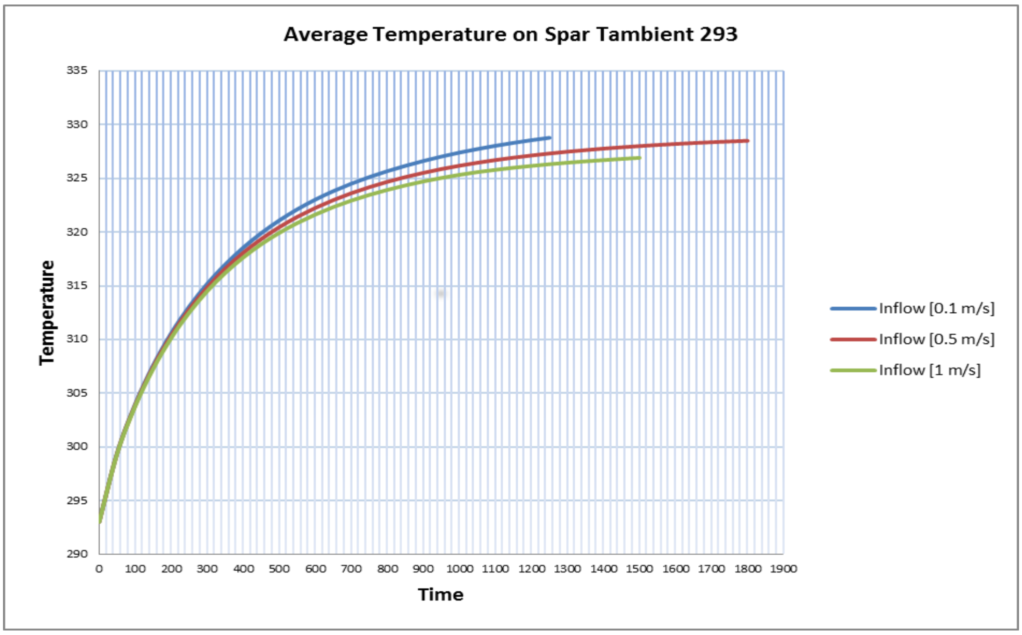


Fig. 10. Temperature history on the front spar for different air inflow values. Time step of some cases became very small, and co-simulations became very time consuming. These simulations are terminated. The simulated time is good enough to predict the steady state situation.

Fig. 10 shows the temperature history of the average temperature of the front spar with an output time step of 50 seconds. For different air inflow velocity values the final temperature approaches a value between 327K and 329 K. Extensive simulations have been carried out to study the ventilation performance of the non-standard enclosure in the wing leading edge. As can be seen in Fig. 10, the effect of varying air inflow on the wing spar temperature is present but is very limited in the composite structure.

The heat transfer inside the leading edge is illustrated in Fig. 11. As shown, the influence of conduction via the top and bottom leading edge surface to the spar is the most visible effect of transferring the heat. The convection effects are present but get suppressed by the conduction effects in the early simulation times. The second dominating effect is by heat transfer between fuel tank and spar which was taken into account via film coefficients for fuel, and ullage and constant sink temperatures.

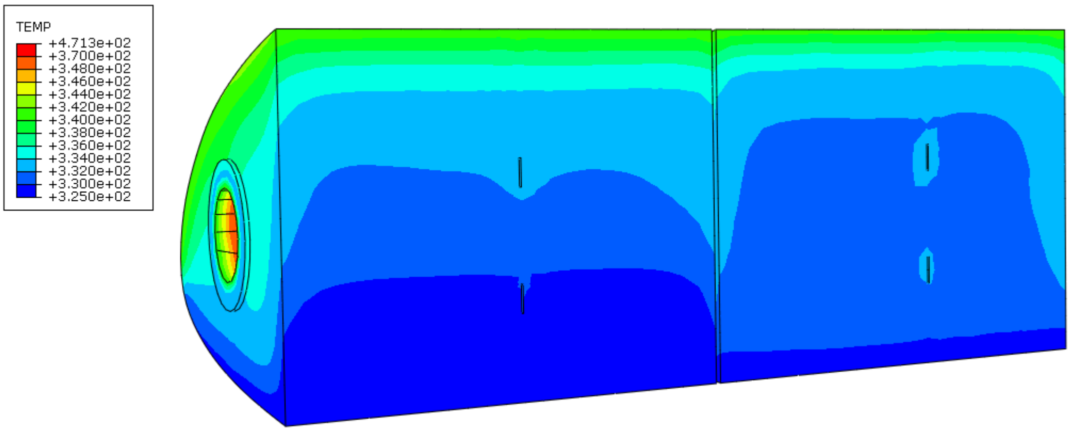


Fig. 11. Dominating effects of heat transfer through conduction via top and bottom surface and suports.

The air next to the bleed pipe heats up via convection and the heat reaches the bleed pipe supports which in turn transfer heat to the spar via conduction. In Fig. 11 the heat transfer via these supports is shown. But the limited contact area and low radiation value of the bleed pipe, due to isolation material, makes the influence of this type of heat transfer insignificant.

It can be concluded that these detailed simulations provide accurate results which can be used as reliable input for the fuel tank flammability analysis.

Acknowledgements

The authors would like to acknowledge the support of the Padova University for providing the initial wall temperatures.

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

Abaqus. (2016). Abaqus documentation. *release 2016*. Dassault Systèmes.

Blades, E.L., Luke, E.A., Kurkchubashe, A.G., Collins, E.M. (2010). A fluid-structure interaction simulation capability using the co-simulation engine. *SIMULIA Customer Conference.* Simulia.

International, S. (1990). *SAE AIR 1168/3 Aerodynamic systems engineering and design.*

Moore, D., Newport, D., Egan, V., Lacarac, V. (2012). Ventilation and internal structure effects on naturally induced flows in a static aircraft wing. *Applied Thermal Engineering*, 49-58.

Summer, S. (2008). *Fuel Tank Flammability Assessment Method User's Manual.* Federal Aviation Administration.

Vale, C., Ringrow, H. (2011). *Family optimized air-vent inlet NACA Duct.* Airbus.

This page is intentionally left blank.

NLR

Anthony Fokkerweg 2
1059 CM Amsterdam, The Netherlands
p) +31 88 511 3113 f) +31 88 511 3210
e) info@nlr.nl i) www.nlr.nl