

Università degli Studi di  
Napoli Federico II  
Facoltà di Ingegneria



Corso di Studi in  
Ingegneria Meccanica

Laurea Magistrale in Ingegneria Meccanica per l'Energia e l'Ambiente  
(Classe delle Lauree Magistrali in Ingegneria Meccanica – L.M. 33)

Tesi di Laurea

Aerothermal analysis of an Aircraft Nacelle in the Framework of a Fully Coupled  
Approach

**Relatori:**

Ch.mo Prof. Ing. Nicola Bianco  
DII - Dip. di Ingegneria Industriale  
Ch.mo Prof. Ing. Marilena Musto  
DII - Dip. di Ingegneria Industriale  
Ch.mo Prof. Ing. Giuseppe Rotondo  
DII - Dip. di Ingegneria Industriale

**Correlatore:**

Ing. Giuseppe Pezzella  
C.I.R.A. - Centro Italiano Ricerche Aerospaziali

**Candidato:**

Gennaro Buonomo  
matr. M65/122

**ABSTRACT**

The integration of a powerplant into an airframe presents challenges in the thermal and ventilation design of the engine nacelle. In fact, high fidelity, fully coupled, “aero-thermal” models must be addressed in the design process.

In this framework, the thesis work reports on a thermo-fluid-dynamic analysis of the flowfield around the aircraft nacelle (Fig. 1) equipped with a small pusher turbine engine, with the purpose of determining the thermal field on nacelle surface.

The Computational Fluid Dynamics (CFD) simulations couple flow and thermal fields, solving the thermal conduction inside the nacelle wall (conjugate heat transfer analysis) due to the presence of the engine hot components, and taking into account nacelle cooling/ventilation system. Both convective and radiative heat transfer are considered in the CFD analysis with proper boundary conditions at engine surface and nacelle wall. Numerical results (Fig. 2) highlight that the radiative heat transfer represent a not negligible contribution to the design of the nacelle, made of composite materials. Geometry analysis has been done by the Rhinoceros 5 CAD, while for the mesh generation has been used Ansys Icem 14. 3-D CFD calculations were performed by solving the RANS equations with the commercial CFD code, Ansys Fluent 14. The density-based flow solver and the standard SST  $k-\omega$  model for turbulence were employed. In addition, the radiative exchange between engine and nacelle was also taken into account, by using the Discrete Ordinate model available in Ansys Fluent 14.

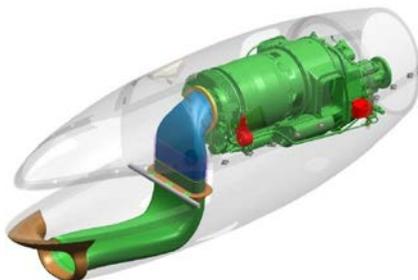


Fig.1- Nacelle overview, both engine and skin.

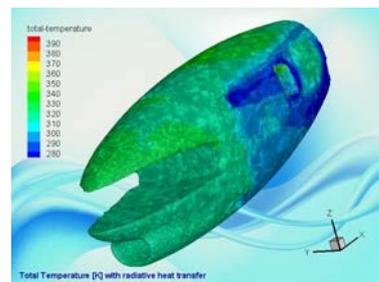


Fig. 2 - Contour of static temperature.