VEGA C : AERODYNAMICS AT SEPARATION OF 1ST STAGE

F. Paglia

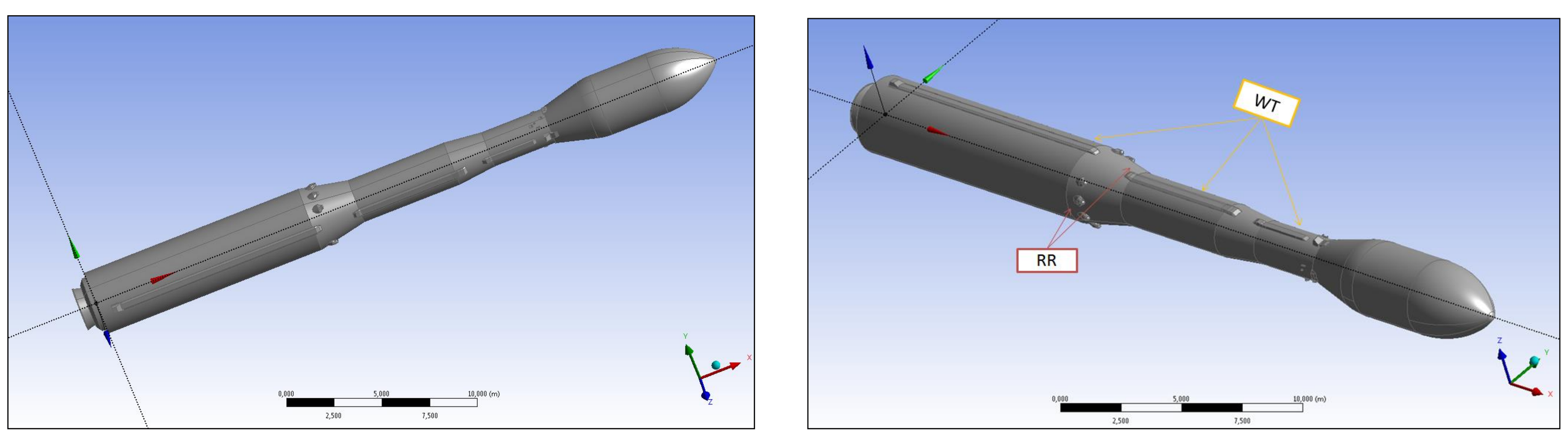
Avio S.p.A. , Colleferro (Rome), Italy

Introduction

The VEga Consolidation and Evolution Programme (VECEP) has entered into force on 21 November 2012, following subscriptions made by participating States at the occasion of the ESA Council meeting at ministerial level in Naples. Following the Scenarios Expert Group sessions hold during summer 2014, a new orientation in the Ariane programmes has been defined, which has impacted the Vega evolution scenario. Indeed it has been decided to develop a common SRM (hereafter named P120C) to be used both as Vega 1st stage SRM and strap-on boosters for the Ariane 6-2 (2 boosters) and Ariane 6-4 (4 boosters) PHH configurations. Furthermore, this change on the 1st stage propulsion has led to suggest the substitution of the Zefiro 23 with a new 2nd stage SRM - the Zefiro 40 - which provides a better staging of the Vega launcher. The new baseline architecture for Vega C has become then: P120C/Z40/Z9/AVUM+. The Vega C launcher will increase by at least 700 kg the nominal Vega performance on its reference mission and provide an enhanced service at a recurring cost not exceeding the Vega second batch procurement cost.
Aim of this work is to present a study of flow field due to retro-rockets impingement during the 1st stage VEGA C separation phase. In particular the main goal of the present work is to understand the physical behavior of the flow field during separation phase by means of CFD analysis

Physical problem

The concept of 1st stage separation assisted by retro rockets is an heritage of previous European launchers, in particular of Ariane family (from version 1 till version 4). The retro motors used by VEGA C are exactly the same of Ariane 4.
The separation of VEGA C 1st stage is made by the pyro-cutting of the interstage 1-2 structure and occurs at the end of the combustion phase of the first solid rocket motor P120C at a relatively low altitude. Due to the residual thrust of P120C and the interaction of the 8 separation solid rocket motors with the external aerodynamic field, it is a critical phase.



Geometrical configuration of VEGA C

Numerical approach

The commercial CFD code ANSYS FLUENT has been used for all the numerical simulations. FLUENT uses a control-volume-based technique for discretization and numerical solution of field equations. The solution method for the conservation equations is based on a coupled implicit algorithm. The convective terms in the equations for momentum and energy are discretized with a second order upwind scheme. A standard k-ε model is used for turbulence. An un-structured mesh of about 20 million of total computational cells has been adopted during numerical simulations.

Facilities

Numerical simulations have been performed using the integrated computational infrastructure ENEAGRID\CRESCO. The availability of powerful computing systems, distributed over wide area and connected by high speed networks, has lead to the development of the concept of computational GRID. The GRID is an attempt to provide a unified approach to heterogeneous computational resources located in distant sites and belonging, in the most general case, to institutions having very different types of activities.

Results

The external aerodynamics that develops around the launcher during the separation manoeuvre is outside the field of classical aerodynamics showing a completely different behavior. The supersonic jet of the RR, flowing towards the opposite direction of the hypersonic mainstream strongly interacts with the external field generating a 3D shock wave which completely changes the aerodynamic of the two separating bodies.

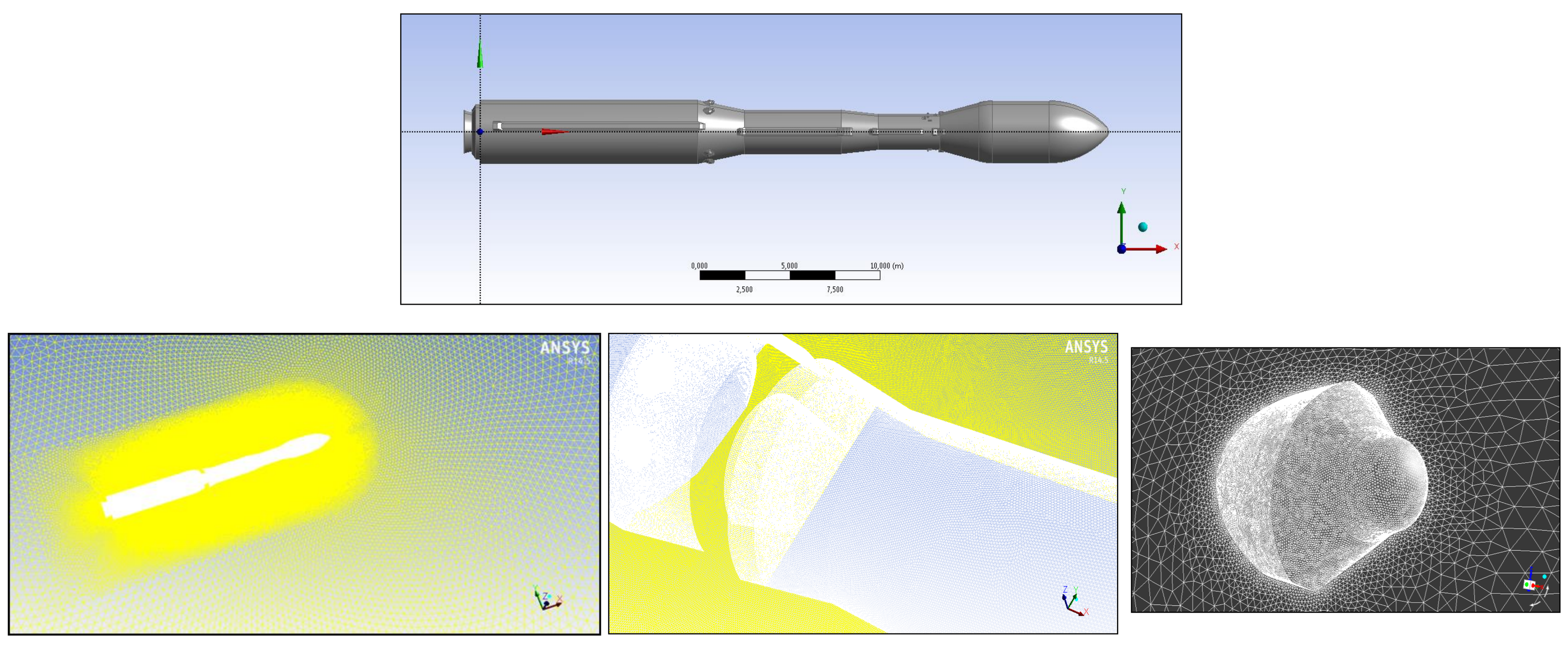
CFD analyses conducted have clearly shown that the flow field around the launcher is strongly affected by the physical behavior of some different zones:

#1 : External not perturbed flow

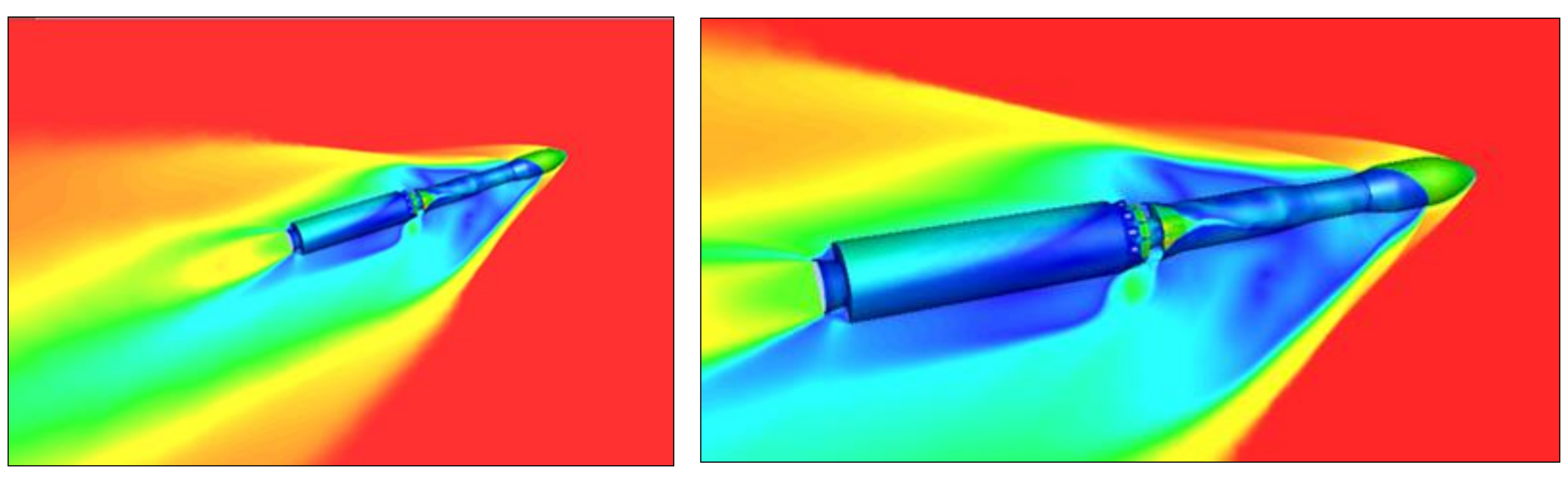
#2 : High mixing region between the shock due to the retro-rockets plume and the shock due to the external flow

#3 : Expansion of the RR exhaust jets along the Zefiro external surface, due to the intrinsic under expansion of the plume at the exit of RR

These zones driving pressure distribution on the external surface of the launcher obviously becomes the main responsible of the overall force and moment values.



Computational domain and Mesh



Mach number contour plot

Conclusions

- A robust CFD approach has been developed in order to understand the physical behavior of the flow field during separation phase 1-2 of VEGA C.
- CFD analyses conducted have shown that the flow field around the launcher is strongly affected by the physical behavior of some different zones, that, driving pressure distribution on the external surface of the launcher, becomes the main responsible of the overall force and moment acting on LV