High mach number drag analysis of a modern lightweight launch vehicle

Ainslie D French¹, Antonio Schettino¹ and Luca Romano²

¹CIRA (Italian Aerospace Research Center), Via Maiorise, Capua (CE), Italy 81043
²Avio s.p.a., Corso Garibaldi, 20, - Colleferro (Rome), Italy, 00034

Corresponding author: a.french@cira.it

Abstract. This paper presents predictions of pressure and frictional drag using on the individual components of the VEGA C Light Launch Vehicle propelled by a solid rocket motor over a hypersonic Mach number range from five to eight at nominal atmospheric conditions associated with the corresponding stage of the ascent trajectory. Both motor-off and motor-on conditions are simulated to isolate the motor-on effects. A series of simulations were also performed in off-nominal conditions by varying the Reynolds number and mass flow rate to examine the effects on drag under more extreme conditions. The results indicate that the differences between the motor-off and motor-on cases in nominal conditions are mainly due to base drag. With the motor-on the base pressure increases thereby reducing the total drag. In the off-nominal conditions, when the Reynolds number is increased there is a reduction in drag, and when the nozzle mass flow rate is decreased there is an increase in drag. Conversely, when the Reynolds number is decreased and the nozzle mass flow rate is increased, the opposite effect occurs. However, for very low Reynolds numbers, an incipient separation can occur on the first stage, which also influences the drag.

1. Introduction

The accurate evaluation of the drag coefficient is crucial to the design of a launch vehicle. For a given configuration, the drag is influenced by the freestream conditions (Mach, Altitude), which are mission dependent. Consequently, when generating the aerodynamic database, the Reynolds effects on drag coefficient must be estimated. Moreover, the nozzle exit plume can significantly affect the drag.

The goal of the present paper is to analyze these effects in detail on the VEGA C Light launch vehicle propelled by a solid rocket motor (SRM) being designed by Avio s.p.a., by conducting a drag breakdown on the various vehicle components as shown in Figure 1. The particular drag components calculated for each geometrical component in this study are the pressure (form) and frictional drag. This paper follows research conducted at CIRA for Avio s.p.a.
In the afterbody region there is an abrupt decrease in the diameter between the end of the first stage and the base and between the base and the nozzle. These discontinuities can cause the separation of the turbulent boundary layer on the rear shoulder of the first stage. In these circumstances a large recirculation region is created downstream of the shoulder and a turbulent shear layer is generated [1].

2. Test matrix and numerical simulation methodology

The drag breakdown calculations were conducted over a range of Mach and Reynolds numbers divided into two phases described in the test matrix using the CIRA software NExT 3.0 (Numerical Experimental Tool) [2] developed in house and the commercial software ANSYS-FLUENT v14.5 [3] as an additional validation of the results obtained over the Mach and Reynolds number ranges considered.

2.1. Test matrix

The test matrix associated with the nominal conditions and far field for the motor on and motor off test cases is shown in Table 1. The Reynolds numbers shown are averages associated with the trajectories.

| Mach number | Reynolds number (x10⁶) | Mass Flow Rate (non-dimensional) |
|-------------|-------------------------|----------------------------------|
| 5.0         | 1.40                    | 1.000                            |
| 5.5         | 0.55                    | 0.975                            |
| 6.0         | 0.09                    | 0.359                            |
| 7.0         | 0.01                    | 0.008                            |
| 8.0         | 0.003                   | 0.0025                           |

The conditions associated with additional simulations for the motor on test case are shown in Table 2, where the mass flow rate (MFR) is modified by changing the SRM chamber pressure.

| Departure from nominal | Reynolds number | SRM pressure | Mach number range |
|------------------------|----------------|--------------|-------------------|
| Re_{max} MFR_{min}     | +20%           | -15%         | 5, 5.5, 6, 7, 8   |
| Re_{min} MFR_{max}     | -95%           | +15%         | 5, 5.5, 6, 7, 8   |
2.2. Numerical simulation methodology

2.2.1. Domain and grid parameters. The calculations were performed on similar fine grids for the motor on and motor off test cases. The grids used in the motor on and motor off calculations are shown in Figure 2 and Figure 3. The grid shown, used with NExT CIRA code, consists of 223760 cells distributed over 48 blocks and was interpreted as an unstructured grid of similar size for the subsequent ANSYS-FLUENT calculations. The grids were generated with the ANSYS® ICEM CFD™ software and satisfy the associated quality criteria.

![Figure 2. Grid used for far field motor on (a) and motor off (b) calculations](image)

In the motor off calculation the nozzle base is closed off as a solid wall as shown in Figure 3.

![Figure 3. Nozzle region of grid used for motor on (a) and motor off (b)](image)

2.2.2. Solver and boundary conditions. The NExT CIRA code solved the 3D compressible Navier-Stokes equations using an explicit scheme for the time integration to approach the steady state on multiblock grids. The convective scheme used second order upwind flux differencing and turbulence was modelled with the standard k – ε turbulence model.
For the ANSYS-FLUENT® calculations the compressible 3D Navier-Stokes equations were solved with the density-based solver using the Advection Upwind Splitting Method (AUSM) second order convective flux solver. Pseudo transient time-stepping was employed combined with the shear stress transport (SST) $k – \omega$ turbulence model.

For the present computations both the free-stream air and the nozzle inlet were modelled as perfect gases, by imposing the physical parameters associated with the vehicle trajectory and the associated motor characteristics for the respective test case. The wall temperature was set to 300 K for all the external surfaces whereas on the internal wall of the nozzle a temperature of 2500 K was imposed.

2.3. Numerical results

2.3.1 Numerical results for phase 1. Comparison of results obtained with NExT for phase 1 for the nominal test cases indicated in Table 1, associated with the engine off/on configurations in terms of lumped drag coefficients are presented from Figure 4 to Figure 7, below.

The pressure and friction contributions are emphasized separately, in order to appreciate their individual contributions to the total drag. From these figures it can clearly be seen that the change of drag coefficient between the “off” and “on” conditions is mainly due to the base drag, which is obviously influenced by the high-pressure plume (Figure 7a). A small difference can also be seen in the frictional drag on the first stage, but only at the higher Mach numbers (Figure 6b).

As a general comment, it should be noted that, as expected, at low Mach and high Reynolds numbers the most significant contribution to the total drag is predominantly due to the fairing, and partially due to the base. However, by increasing the Mach number, i.e. reducing the Reynolds number (which is only associated with the change in trajectory, altitude and speed, and not with the mathematical relationship between Mach No. and Reynolds No.), the friction contribution also becomes significant, being of the same order of magnitude as the pressure contribution.

![Figure 4. Comparison of total CD for engine off/on (a) and Mach 8 isolines (b)](image)

At low Reynold numbers associated with high altitude, decreasing density and high Mach numbers the effect of the high-pressure plume in the engine on condition is particularly evident, see Figure 4b.
2.3.2 Numerical results for phase 2. Comparison of results obtained with NExT for phase 2 associated with maximum/minimum Reynolds number and minimum/maximum SRM chamber pressure, as indicated in Table 2, with the nominal values for the engine on configuration presented in terms of lumped drag coefficients are shown from Figure 8 to Figure 11.

For the first condition, associated with an increase of Reynolds number of 20% and a 15% decrease of the motor pressure, two opposite effects were expected. On the one hand, the higher Reynolds number
causes a reduction of the friction, and consequently of total drag, but on the other hand, the lower pressure close to the base causes an increase in drag. However, from Figure 11 it can be seen that the base drag is substantially unchanged, and therefore the most significant global effect is a decrease in total drag, of 5% at higher Mach numbers, due to the variation in Reynolds number.

Figure 8. Comparison of total CD for $\text{Re}_{\text{max/min}} \text{ SRM Pres}_{\text{min/max}}$ (a) and Mach 8 isolines (b)

Figure 9. Comparison of CD for $\text{Re}_{\text{max/min}} \text{ SRM Pres}_{\text{min/max}}$ for fairing (a) and second stage (b)

Figure 10. Comparison of CD for $\text{Re}_{\text{max/min}} \text{ SRM Pres}_{\text{min/max}}$ for inter-stage (a) and first stage (b)
Figure 11. Comparison of CD for Re_{max/min} SRM Pres_{min/max} for base (a) and nozzle (b)

For the second case, associated with a decrease in Reynolds number of 95% and a 15% increase of the motor pressure, the most significant effect is a strong increase in the total drag at high Mach numbers, as expected. However, it is worth noting that local reduction of the drag in the base region, which for the lower Mach numbers is due to the increase of motor pressure, while for the higher Mach numbers the drag reduction is due to the negative contribution of the friction, caused by the flow separation on the first stage, as shown in Figure 8b.

The crucial aspect to the creation of different mean flow and turbulent flow topologies in the aft region of the launcher depends on whether or not flow reattachment occurs on the outer surface of the motor nozzle.

A significant reduction in the Reynolds number equates to a similar reduction in density which can be associated with a much higher altitude of the launcher. Combined with a simultaneous increase in nozzle mass flow rate synonymous with a more powerful under-expanded plume, more extreme conditions of recirculating flow in the base and nozzle region are generated together with a strong displacement of the outer flow due to the stronger plume which both seek to prevent flow reattachment at the nozzle base and produce the dramatic separation characterized in this analysis.

Figure 12. Comparison of global CD NExT & FLUENT (a) and Mach 8 isolines (b) for engine on

2.3.3 Comparison of results between NExT and ANSYS-FLUENT®. Comparison of results obtained with NExT and ANSYS-FLUENT® for the nominal test cases indicated in Table 1, associated with
the engine on configuration in terms of lumped drag coefficients are presented in Figure 12 and Figure 13 where only the components associated with the major differences in drag predictions are reported. The two codes compare fairly well with small differences in the frictional drag mainly on the fairing and first stage. This is probably due to the different turbulent models used.

Use of the standard $k-\varepsilon$ and $k-\omega$ turbulence models could also explain the difference in the Mach number distributions predicted by the two codes in simulating the high-pressure plume, a region of high turbulence, as shown in Figure 12b.

![Comparison of CD NExT & FLUENT for fairing (a) and first stage (b) for engine on](image)

**Figure 13.** Comparison of CD NExT & FLUENT for fairing (a) and first stage (b) for engine on

### 3. Conclusions

The differences between motor-off and motor-on cases, in nominal conditions, are mainly due to base drag. With the motor on the base pressure increases thereby reducing the total drag.

In the off nominal, conditions the Reynolds number increase causes a reduction in drag, while the decrease of the mass flow rate causes an increase. However, the Reynolds number effect is predominant, leading to an increase of the total drag of up to five percent. Conversely, when the Reynolds number is decreased and the mass flow rate is increased a very strong increase in the total drag was predicted, especially at high Mach numbers.

These latter conditions are equivalent to flight at even higher altitudes and a more powerful supersonic under-expanded flow exiting the nozzle than in the nominal test case, giving rise to a highly expanded plume which tends to strongly displace the outer flow thus preventing reattachment on the nozzle outer surface and creating the strong separation modelled on the first stage.

Comparison of the two codes, NExT and ANSYS-FLUENT®, indicates fairly good agreement for the engine on test case. Small differences in frictional drag are probably due to the different turbulence models used.

### Acknowledgments

Thanks to Avio s.p.a. for the funding of this research work to CIRA under the project VEGA-L Work Package No. D2230A.

### References

[1] Schreyer A M 2020 “Flow structure in the wake of a space-launcher model with propulsive jet simulations” CEAS Space Journal p 367-383.

[2] Ranuzzi G, Cutrone L, Cardillo D and Invigorito M 2016 “Numerical Investigation of Rocket Engine Combusting Flowfields” AIAA-2016-2147, 54th AIAA Aerospace Meeting, AIAA SciTech, San Diego, CA, USA.

[3] ANSYS FLUENT User’s Guide, release 14.5, ANSYS Inc., Canonsburg, Pennsylvania, USA.