Results in Engineering 4 (2019) 100056

Contents lists available at ScienceDirect

Results in Engineering

journal homepage: www.editorialmanager.com/rineng/Default.aspx

Rocket plume URANS simulation using OpenFOAM

Mara S. Escarti-Guillem, Sergio Hoyas [*], Luis M. García-Raffi

Instituto Universitario de Matematica Pura y Aplicada, Universitat Politècnica de València, 46022 València, Spain

A R T I C L E I N F O

Keywords:
OpenFOAM CFD
Launch rocket
Exhaust gases
Supersonic impingement

A B S T R A C T

A key aspect in the launch environment of a space vehicle is the generation and propagation of noise, due to its effect in the payload. Obtaining experimental measures is an extremely difficult, so CFD techniques appear as a useful tool to estimate both the spatial location of noise sources and their spectral power. In this paper, a numerical simulation has been carried out to simulate the supersonic impingement of the plume jet generated by the engine and the corresponding acoustic load generated in the launch platform of the VEGA rocket using a URANS model in OpenFOAM.

When a rocket is launched, a high temperature jet plume is exhausted at supersonic speed. The plume jet generates a large acoustic load which in turn excites undesirable structural vibrations. Therefore, there is a growing interest in understanding the noise generation and propagation phenomena. There are three ways to obtain information about acoustic loads: theoretical calculation, experimental laboratory studies on models, and flight measurements in vehicles. Due to the hostile environment that takes place during launching, characterization of the physical conditions at the launch phase through field measurement is extremely difficult. A widely used empirical method defined in NASA SP-8072 [1], makes use of microphone phase array to locate the sound source distribution. However, the accuracy of prediction demonstrated of this methods is insufficient.

A tool that has the potential to replace the empirical experiments and speed up the design stage is Computational Fluid Dynamics. CFD can provide high fidelity physic predictions modeling the flow behavior in the real scale. In this paper, an URANS simulation of a rocket exhaust in the launch area is performed. Specifically, the VEGA launch vehicle is analyzed at zero altitude.

All the simulations of this paper were carried out using OpenFOAM v6 software [2] which is organized in a set of C++ modules. The main advantages of this open software tool, apart its reliability, is that it allows access to the source code. This gives researchers the option to develop different packages, which offer a wide range of possibilities. It has to be noted that the simulations are carried using parallel decomposition, which is made automatically.

The physical problem assessed is the impingement of the supersonic exhaust plume with the plume deflector as well as the propagation of the exhaust gas jet at the exhaust duct. In supersonic flows the appearance of discontinuities, such as shock waves, affects negatively the numerical stability and the accuracy of the solver. Therefore, the solver and the numerical schemes have to be properly implemented. The solver used is *rhoPimpleCentralFoam* a transient and compressible solver developed by Kraposhin et al. [3]. It is an hybrid method which combines Godunov-type fluxes with operator-splitting techniques for the closure problem similar to PIMPLE. Furthermore, the divergence, the gradient and the laplacian terms are discretised with Gauss linear numerical schemes while the interpolation scheme used is Van Leer.

In order to describe the flow we have to use the $k-\omega$SST turbulence model defined by Menter [4]. This model is perfect for our study because it combines the $k-\omega$ model for boundary layer problems and the $k-\varepsilon$ in the free-stream. Moreover, it has been used by Gusman and Housman [5] to simulate launch vehicle plumes.

The 3D computational domain enclose the geometry representation of the launch platform and the VEGA rocket [6]. Therefore, in order to capture the flow in all the geometry and to avoid the influence of the domain size on the results, the domain is set to 150 × 150 m and 200 m high. The computational mesh was generated with the tool snappyHexMesh and consists of $1e7$ cells, which are mainly hexahedrical elements. Larger meshes have also been used, but this one presents the best quality/cpu-time ratio. Fig. 1 shows a snapshot of the geometry in color grey and a slice of the inner mesh. The element size grows rapidly from

---

[*] Corresponding author.
E-mail addresses: maesgui2@etsid.upv.es (M.S. Escarti-Guillem), serhocal@mot.upv.es (S. Hoyas), lmgarcia@mat.upv.es (L.M. García-Raffi).

https://doi.org/10.1016/j.rineng.2019.100056
Received 25 October 2019; Accepted 30 October 2019

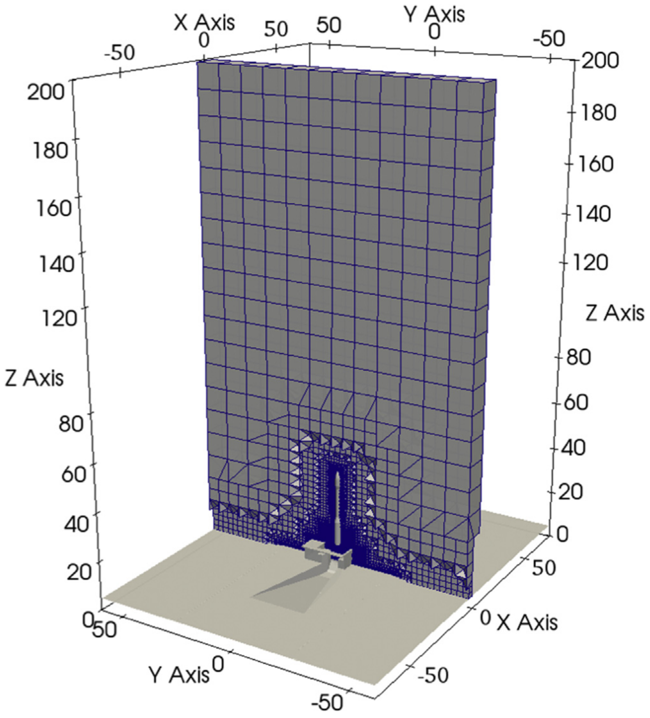

Fig. 1. Launch Vega platform and rocket.

Table 1
Boundary conditions in OpenFOAM.

	Inlet	Outlet	Wall
U (m/s)	fixedValue 2713	zeroGradient	noSlip
p (Pa)	totalPressure 9.5e6	waveTransmissive	zeroGradient
T (k)	fixedValue 1699.89	zeroGradient	zeroGradient
K (m^2/s)	uniform 9746.96	zeroGradient	kqRWallFunction 1e-15
ω (s^{-1})	fixedValue 1.3712e6	zeroGradient	omegaWallFunction 5.73e7

the impingement area to the far field. The smallest elements are those placed in the region where the jet develops and impact with the plume deflector along with the elements in the duct wall.

With regard to the boundary conditions, they are listed in Table 1. The inlet conditions of the flow are applied at the end of the nozzle of the VEGA rocket. This supersonic characteristics have been calculated with the isentropic flow equations through a laval nozzle and the information found in Refs. [7,8]. At the outlet, Neumann boundary conditions are applied for all the parameters except for the pressure where a non-reflective condition has been used. Finally at the walls of the exhaust duct, wall functions are used to define the transport variables of the model.

ParaView v5.7 [9] has been used to perform the post processing of results. Analysis of the flow behaviour to understand the noise generation was the main objective.

Fig. 2 shows the velocity contours and the flow streamlines after exhaust gases have impacted the plume deflector and are spreading along the exhaust duct. The inner core develops at the highest velocity which diminish due to the impact with the deflector tip. Then, the flow accelerates while it propagates along the deflector to be slowed down again at the change of curvature between the deflector and the ground. Because of the exhaust jet propagation along the duct and upwards, a recirculation bubble appears (see Fig. 3).

The identification of noise sources has started with the study of shock waves as they are one of the main phenomena of noise generation. An approximation to the Schlieren images (Fig. 3) have been used to distinguish the different shock waves that appear, using their effect in the density of the flow.

The shock waves observed are those due to the boundary of the jet and the flow moving on the platform; the detached wave on the deflector and the oblique shock wave in the change of curvature of the ramp. The presence of these two shock waves cause the flow decelerations that can be seen on Fig. 2. Notice that the detached shock wave is produced due to the deformation of the normal shock wave generated by the advance of the supersonic jet. This two shock waves are the main contributors to the noise generated by shock waves.

As conclusions, this work presents a CFD simulation that model the highly hostile environment of a rocket launch. It has been possible to provide closer and more precise information regarding the launching of the rocket and the main sound sources. OpenFOAM has been revealed as a very good tool for simulating this kind of flow.

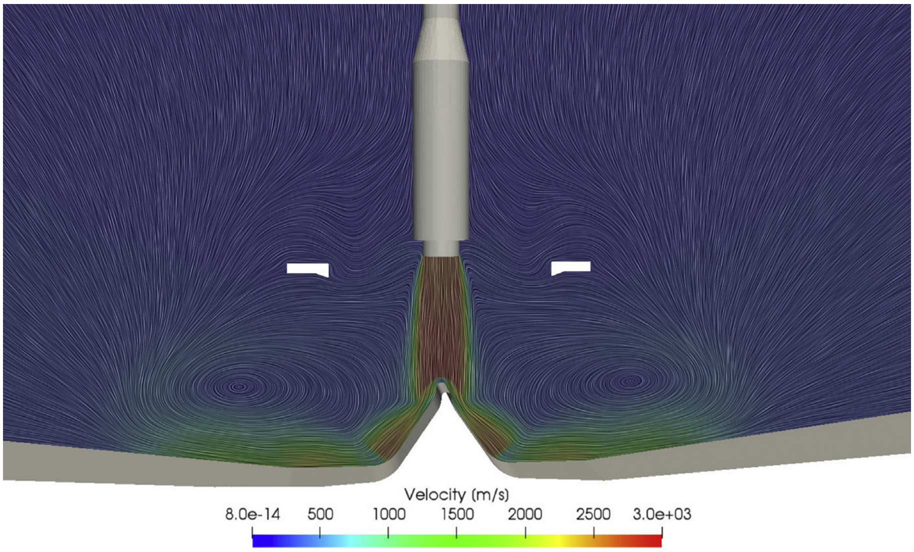

Fig. 2. Velocity contours and flow streamlines.

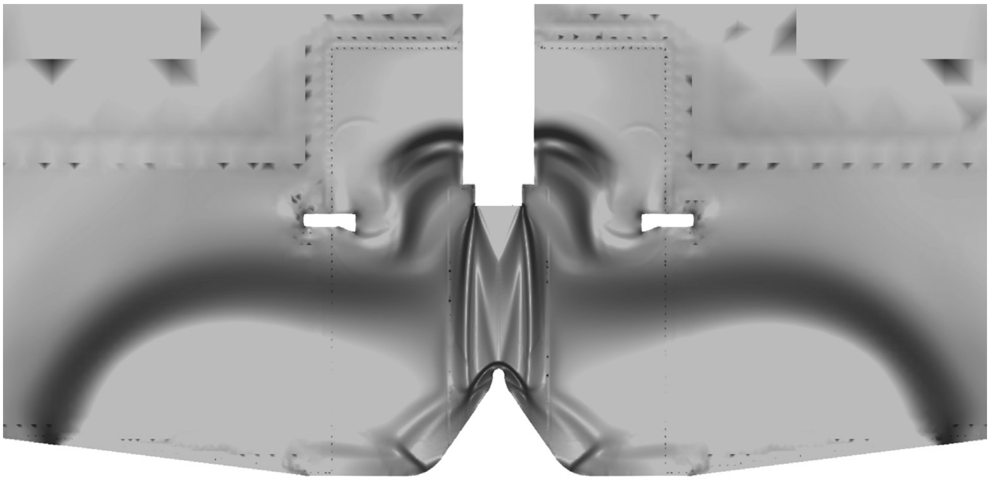

Fig. 3. Synthetic images of Schlieren for the representation of shock waves at the evacuation duct.

Conflict of interest

The authors declare no conflict of interest.

Acknowledgment

This work was partially funded by MINECO / FEDER, under project TI2018-102256-B-I00.

References

[1] R.H. Lyon, Acoustic loads generated by the propulsion system, National Aeronautics And Apsce Administration, 1971.
[2] www.OpenFOAM.com. (Accessed 30 September 2019).
[3] M. Kraposhin, M. Banholzer, M. Pfitzner, I. Marchevsky, A hybrid pressure-based solver for nonideal single-phase fluid flows at all speeds, Int. J. Numer. Methods Fluids 88 (2018) 79–99.
[4] F.R. Menter, Zonal two qquation k-w, turbulence models for aerodynamic flows, in: 24th Fluid Dynamics Conference, 1993.
[5] M. Gusman, J. Housman, Best practices for cfd simulations of launch vehicle ascent with plumes – overflow perspective, in: 49th AIAA Aerospace Sciences Meeting, 2011.
[6] Sonic Crystals For Noise Reduction At The Launch Pad", ESA ITT 1-7094 (ITI).
[7] Ariane Space, VEGA User's Manual, 2014.
[8] M. Calabro, E. Gizzi, N. Wingborg, Y. Batonneau, R. Beauchet, C. Kappenste, Propellant Requirements and Future Demands, 2016.
[9] www.ParaView.org. (Accessed 30 September 2019).