"HENRI COANDA"
AIR FORCE ACADEMY
ROMANIA

GENERAL M.R. STEFANIK
ARMED FORCES ACADEMY
SLOVAK REPUBLIC

INTERNATIONAL CONFERENCE of SCIENTIFIC PAPER
AFASES 2015
Brasov, 28-30 May 2015

THERMO-GAS DYNAMIC ANALYSIS OF UPPER-STAGE ROCKET ENGINE NOZZLE

Mihai Mihaila-Andres*, Paul Virgil Rosu*

*Institute for Theoretical & Experimental Analysis of Aeronautical Structures STRAERO, Bucharest, Romania

Abstract: *A numerical model for the analysis of thermo-gas dynamics of rocket engine exhaust gases has been developed and described in the present study. This model corresponds to liquid propellant rocket engines designed for upper-stages of multi-stage rockets. The work is focused on the simulation of the rocket plume of the Vinci cryogenic rocket engine. Using available data, an analytical study of the convergent divergent flow was made in order to obtain more sensitive information about the physical properties of the exhaust gases. This new data was used as input for the numerical model and the solution obtained was in good agreement with experimental data both qualitatively and quantitatively.*

Keywords: *rocket engines, exhaust gases, propulsion, liquid propellant*

1. INTRODUCTION

In general, for a multi-stage rocket, the first stage engine should have high thrust because it is the thrust that determines the mass that can be accelerated for a given quantity of propellant. The rest of the stage engines should have high specific impulse (or exhaust velocity) since this will determine the ultimate velocity to which a mass can be accelerated for a given quantity of propellant..

When it comes to the propellants used to create thrust or specific impulse, two types are generally utilized: solid propellants and liquid propellants. The solid propellant rocket engines are used mainly for small and medium launchers, as simple and reliable last stages for orbital launchers or as strap-on boosters for large launchers. The main advantages of the solid propellants are the storage capacity, the low risk handling and the dismissal of a propellant delivery system but there are also two major disadvantages: the engine cannot be controlled once started and the specific impulse is rather low because of the low chemical energy of the solid propellants. The liquid propellants on the other side have the advantage of high specific impulse and the possibility of controlling the flow using a delivery system [1]. The disadvantages are the storage issues and the high risk of handling.

The research presented here is focused on the liquid propellant engines used in the upper-stages of large launchers, more precisely on the thermo-gas dynamics of the exhaust nozzle of the Vinci cryogenic upper-stage engine of Ariane 5. This engine weights 550kg and has an exhaust velocity of 4650m/s and a thrust of 180kN thanks to an expansion ratio of 240 achieved by a nozzle extension deployed after separation of the main stage [2].

The propellant used is a 5.8:1 mixture ratio of liquid-oxygen liquid-hydrogen which creates a pressure of 60bar in the combustion chamber and can deliver a specific impulse of 465s at an exhaust gas temperature of 3500K for the deployed nozzle exit diameter of 2.15m.

2. THERMODYNAMICS OF VINCI ROCKET ENGINE NOZZLE

In order to determine the performance parameters of the Vinci engine to be used in the subsequent numerical simulation of the exhaust gases we need to start from the thrust equation

$$F = \dot{m} u_e + p_e A_e - p_a A_e \quad (1)$$

In this relation F is the rocket thrust, $\dot{m}$ is the mass flow rate through the nozzle, u_e is the exhaust velocity, p_a is the atmospheric pressure, p_e is the pressure at the exit plane of the nozzle and A_e is the area of this exit plane.

Assuming isentropic conditions and the exhaust gas as a perfect gas, the exhaust velocity can be derived by equating the kinetic energy of the exhaust gas to the change in enthalpy of the gas as it cools and expands through the nozzle. Thus, the exhaust velocity will be

$$u_e = \sqrt{2 c_p (T_c - T_e)} \quad (2)$$

where c_p is the specific heat at constant pressure, T_c is the temperature of the gas in the combustion chamber and T_e is the temperature of the gas at the exit plane of the nozzle.

Since the thrust equation already contains the exhaust pressure, it is convenient to express the exhaust conditions in terms of this pressure. Under the same assumption of isentropic expansion, the temperatures in the combustion chamber T_c and the temperature at the exit plane T_e can be related by the following equation for isentropic processes

$$\frac{T_e}{T_c} = \left(\frac{p_e}{p_c}\right)^{\frac{\gamma-1}{\gamma}} \quad (3)$$

The index γ is the ratio of the specific heat of exhaust gases at constant pressure to that at constant volume. Its value has a significant impact on the resulting thrust. For rocket exhaust gases at high temperature, a typical value would be about 1.2. Using the data presented in the introduction about the Vinci engine, the index γ can be computed in this case once we write the thrust equation in a suitable form.

The specific heat c_p can also be written as a function of this index and the molecular weight of exhaust gases

$$c_p = \frac{\gamma}{\gamma-1}\frac{R}{M} \quad (4)$$

where R is the universal gas constant and M is the molecular weight of the exhaust gases. Knowing the exhaust velocity and the LOx-LH_2 mixture ratio, one can estimate the molecular weight and the temperature of the exhaust gases using the Figure 1.

Substituting for T_e and c_p, the exhaust velocity can be expressed now as

$$u_e = \sqrt{2\frac{\gamma}{\gamma-1}\frac{RT_c}{M}\left[1-\left(\frac{p_e}{p_c}\right)^{\frac{\gamma-1}{\gamma}}\right]} \quad (5)$$

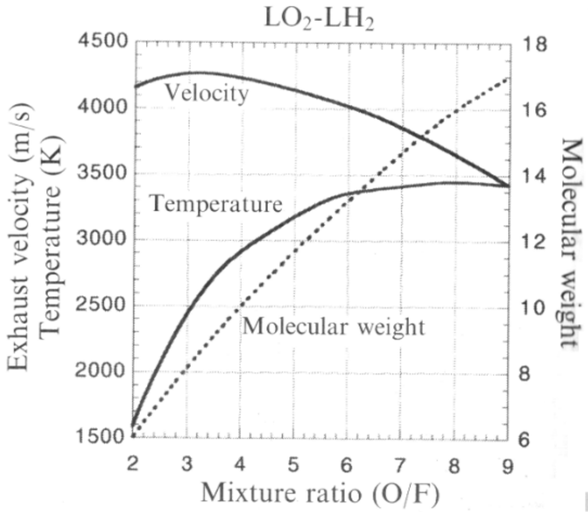


Figure 1. Variation of exhaust velocity, temperature and molecular weight for LOx-LH2 propellant

The mass flow rate $\dot{m}$ can be expressed as a function of the density, velocity and cross-sectional area at any particular point:

$$\dot{m} = \rho u A$$

Using the expression of the exhaust velocity to derive the velocity at any particular point, the mass flow rate can be written as

$$\dot{m} = \rho A \sqrt{2\frac{\gamma}{\gamma-1}\frac{RT_c}{M}\left[1-\left(\frac{p}{p_c}\right)^{\frac{\gamma-1}{\gamma}}\right]} \quad (6)$$

In the above relation the density ρ is unknown and it needs to be expressed in terms of known parameters. Using the assumption of perfect gas and the isentropic expansion, the density at any particular point can be written as a function of combustion chamber parameters

$$\rho = \frac{p_c M}{R T_c}\left(\frac{p}{p_c}\right)^{\frac{1}{\gamma}} \tag{7}$$

Substituting in the mass flow rate equation, the following mass flow rate per unit cross-sectional area of the nozzle is obtained

$$\frac{\dot{m}}{A} = p_c\sqrt{2\frac{\gamma}{\gamma-1}\frac{M}{R T_c}\left(\frac{p}{p_c}\right)^{\frac{2}{\gamma}}\left[1-\left(\frac{p}{p_c}\right)^{\frac{\gamma-1}{\gamma}}\right]} \tag{8}$$

Differentiating the above relation, the peak value, which occurs at the throat of the nozzle, can be determined

$$\frac{\dot{m}}{A_t} = p_c\sqrt{\gamma\left(\frac{2}{\gamma+1}\right)^{\frac{\gamma+1}{\gamma-1}}\frac{M}{R T_c}} \tag{9}$$

Thus the mass flow rate can be determined mainly by the throat area A_t, the pressure and the temperature in the combustion chamber.

Substituting now the expressions for the exhaust velocity u_e and the mass flow rate $\dot{m}$, the thrust equation can be written as

$$F = p_c A_t\sqrt{\frac{2\gamma^2}{\gamma-1}\left(\frac{2}{\gamma+1}\right)^{\frac{\gamma+1}{\gamma-1}}\left[1-\left(\frac{p_e}{p_c}\right)^{\frac{\gamma-1}{\gamma}}\right]} + \tag{10}$$

$$+ p_e A_e - p_a A_e$$

Using the relations presented above and the known data about the Vinci engine we have determined the density, the molecular weight M, the ratio of the specific heats γ, the throat cross-sectional area A_t and the mass flow rate $\dot{m}$ of the exhaust gases which are needed to perform the numerical analysis of the exhaust gases [3].

3. NUMERICAL ANALYSIS OF EXHAUST GASES

The numerical model of the exhaust gas was computed using the CFD software ANSYS Fluent.

Using a structured multi-block strategy, the fluid domain has been meshed in the ICEMCFD meshing tool. The mesh was tuned with O-grid blocks close to the walls of the nozzle and the environment has been modeled as a cylinder. The final structured mesh had 2M hexahedral cells in 8 blocks.

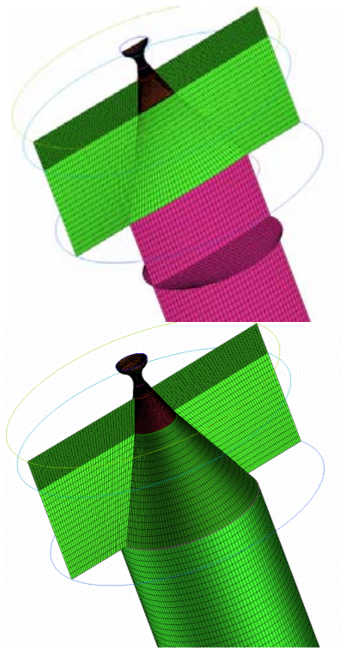


Figure 2. 3D multi-block structured mesh of the computational fluid domain

ANSYS Fluent has implemented two numerical methods for the computation of fluid flow solutions. One is a pressure-based solver and the other is a density-based solver. Both solvers can be used for a wide-ranging variety of flows but since the flow modeled here is a compressible and high speed flow, the chosen solver was density-based which solves the governing equations of continuity, momentum, energy and species transport simultaneously. Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained. In each iteration the fluid properties are updated based on the current solution, the continuity, momentum, and (where appropriate) energy and species equations are solved simultaneously and the equations for turbulence are solved sequentially using the previously updated values of the other variables. The iterations continue until the convergence criteria are met.

The coupled governing equations were linearized implicitly with respect to all dependent variables in the set, specifically for a given variable, the unknown value in each cell is computed using a relation that includes both existing and unknown values from neighboring cells. This implicit formulation gives a system of linear equations for each cell in the domain.

The flow was modeled as viscous transport of species with the energy (heat transfer) equation enabled since the exhaust gases temperatures are essential to our simulation.

The effects of turbulent fluctuations of velocities and scalar quantities were modeled using the 2-equations k-eps “realizable” turbulence model which diverges from the standard k-eps model by certain mathematical constrains on the Reynolds stresses, consistent with the physics of turbulent flow. The near-wall modeling of the eps-equation uses the Enhanced Wall Treatment method which combines a two-layer model with enhanced wall functions. This choice of near-wall treatment reduces the computational time by allowing the use of coarse meshes with a y^+ greater than 1.

For the air/exhaust gases mixture transport simulation, a mixture of two non-reacting species was defined. The properties are defined in Table 1 where the exhaust gases properties (density, molecular weight and specific heat) were obtained from the analytical thermodynamics equations of the Vinci rocket engine described in the beginning of the text.

A pressure-inlet boundary condition was imposed at the combustion chamber/nozzle interface. The mixture has here a 1:0 ratio exhaust gases/air at a gauge total pressure of 60bar and 3500°K. The turbulence properties were imposed in terms of 1% turbulence intensity and 0.1 turbulent viscosity ratio. The same pressure-inlet boundary condition was imposed on the outer areas of the modeled environment. The mixture has here a 0:1 ratio of exhaust gases/air at a gauge pressure of 1atm and at a total temperature of 300°K. The turbulence properties were imposed in terms of turbulence intensity of 5% and turbulent viscosity ratio of 10. The outflow area was chosen far enough from the nozzle so that we could impose a pressure-outlet condition with standard atmosphere conditions at sea level (1atm and 288°K).

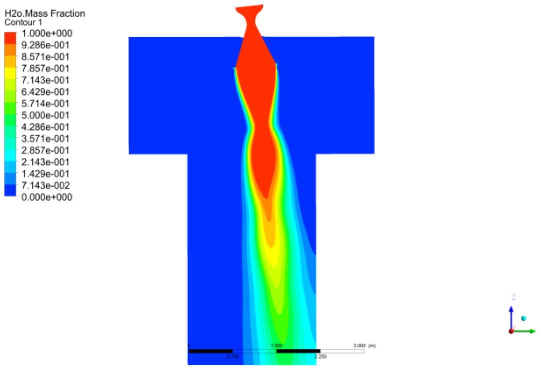
In order to accelerate the flow convergence, an initial “guess” solution was used at the start of the computations based on the Full Multigrid initialization.

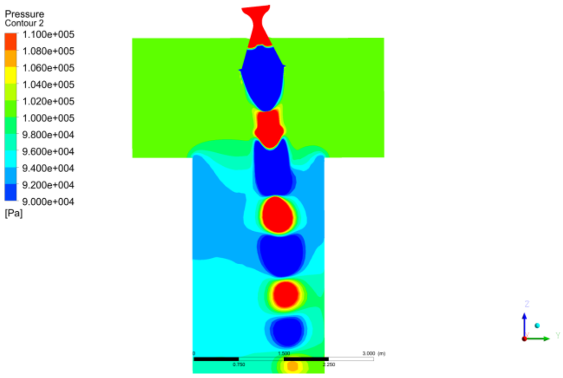
During the solution process we monitored the convergence by plotting the residuals and the total mass imbalance between the inlet of the domain and the outflow.

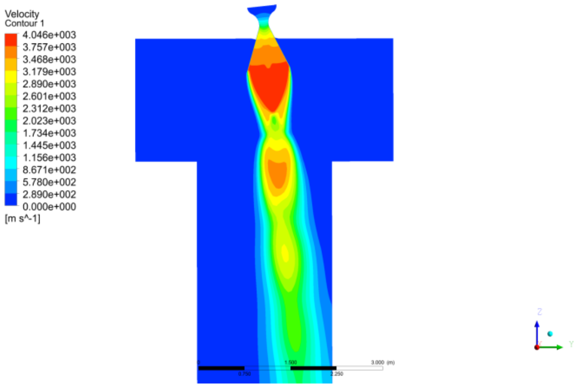
Table 1 Air/Exhaust gases Mixture properties

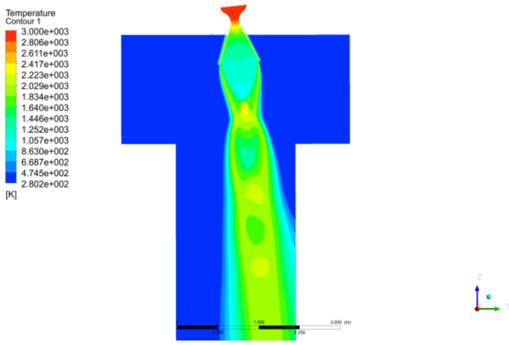
Properties/Species	air	Exhaust gases	Mixture
Density (kg/m^3)	ideal-gas	ideal-gas	ideal-gas
Specific Heat (J/kg-K)	1006.43	4060	mixing-law
Thermal Conductivity (W/m-K)	0.0242	0.0261	mass-weighted-mixing-law
Viscosity (kg/m-s)	1.7894e-05	1.34e-05	mass-weighted-mixing-law
Molecular Weight (kg/kmol)	28.966	12.45	mass-fractions

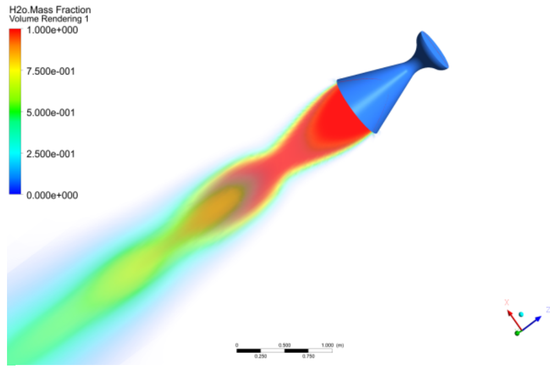
Mass Diffusivity (m^2/s)	-	-	kinetic-theory
Thermal Diff. Coef. (kg/m-s)	-	-	kinetic-theory

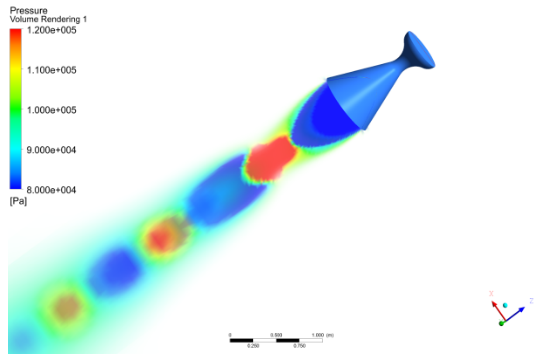


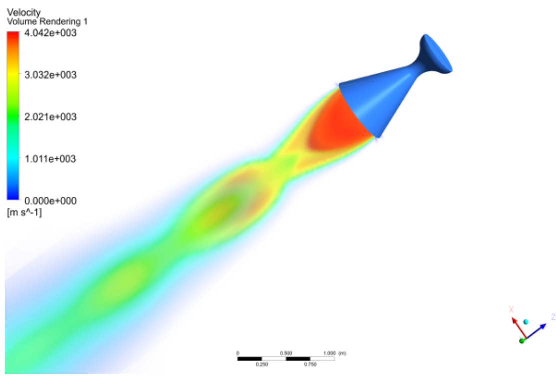


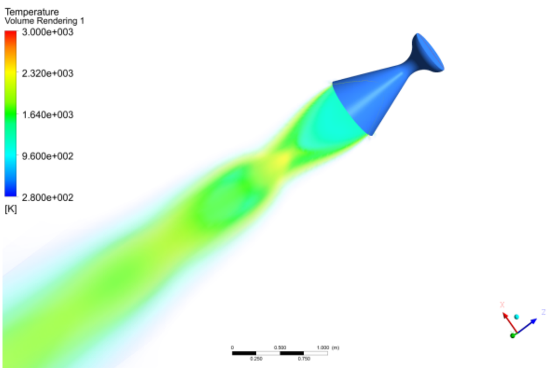












The converged solution is presented in the figures above where the rocket plume is very well defined. In the pressure distribution images one can notice the three shock diamonds that appear in the exhaust plume due to the slightly over-expanded supersonic flow at the nozzle exit [4]. The same behavior can be observed in the combustion chamber of a turbojet engine [5].

These are due to cyclic variations in the plume pressure relative to ambient creating shock waves that form the “Mach disks” [6,7]. At each shock diamond, the flow becomes compressed enough that it expands developing the expansion fan (which is a set of expansion waves). After the flow expands enough so that its pressure is again below ambient, the expansion fan reflects off the contact discontinuity creating the compression fan. The pattern of disks repeats itself three times before the turbulent shear at the contact discontinuity causes the wave pattern to dissipate. The corresponding increase and decrease in temperature can be observed in the contour plot of temperatures of the exhaust gas. In terms of exhaust gases velocity, the contour plot of velocities shows an exit velocity of gases at 4046m/s which has an error of 14% relative to the expected exit velocity of 4650m/s. The error can be attributed to a number of approximations we made in our numerical model (pseudo-transient analysis, estimated turbulence values, species transport, mesh quality, etc.).

4. CONCLUSIONS

In the present work the flow of exhaust gases from a liquid propellant rocket engine have been numerically modeled. In order to perform the appropriate simulations, the combustion chamber and the nozzle were the subject of an analytical study. Using known data about the Vinci cryogenic rocket engine we calculated the characteristics of the exhaust gases (molecular mass, specific heat and the mass flow rate) which were used to complete the boundary conditions for the numerical model. The solution for the simulation was calculated using the CFD software ANSYS where the flow was modeled as a pseudo-transient, turbulent, species transport, energy enabled flow. The results proved the feasibility of the complex model. Qualitatively we obtained the expected rocket plume and we were able to identify its characteristics. Quantitatively, the errors were acceptable for the degree of approximation introduced by the numerical model but opens new scopes for future studies. The mesh influence on precision, a better turbulence model and more accurate estimation of turbulence properties, a full transient analysis and a more complex multiphase model are all possible starting points for more accurate future simulations of rocket exhaust gases flow.

REFERENCES

1. Mattingly J.D., Ohain H, *Elements of Propulsion: Gas Turbines and Rockets*, AIAA Education Series, Reston, Virginia (2006).

2. Sutton G.P., Biblarz O., *Rocket Propulsion Elements*, John Willey & Sons, Inc., New York (2001).

3. Hill, P., Peterson, C., *Mechanics and thermodynamics of propulsion*, Addison-Wesley Publishing Company, (2010).

4. Orlowska, M., Panas, A.J., Recko, K., Zyluk, A., *Numerical Modelling Of The Thermo-Mechanical Response Of A Rocket Motor To Exhaust Gases Load*, Journal Of Theoretical And Applied Mechanics, 52, 3, pp. 803-814, Warsaw (2014).

5. Rotaru C., Andres-Mihaila M., Matei P. G., *"An Extended Combustion Model for the Aircraft Turbojet Engine"*, International Journal of Turbo & Jet Engines, Volume 31, Issue 3, Pages 229-237 (2014).

6. Pandey, K. M. and Yadav, S. K., *CFD analysis of a rocket nozzle with two inlets at Mach 2.1*, Journal of Environmental Research and Development, Vol. 5 No. 2 (2010).

7. Georgiadis, N.J., Dalbello,T.W., Trefny, C.J. and Johns, A.L., *Aerodynamic Design and Analysis of High Performance Nozzle for Mach 4 Accelerator Vehicles*, 44th AIAA Aerospace Sciences Meeting & Exhibit, Reno, Nevada (2006).