Optimal aerodynamic design of scramjet facility nozzles

Enrico Degregori, and Michele Ferlauto

Citation: AIP Conference Proceedings **1978**, 470114 (2018); doi: 10.1063/1.5048596
View online: https://doi.org/10.1063/1.5048596
View Table of Contents: http://aip.scitation.org/toc/apc/1978/1
Published by the American Institute of Physics

Optimal Aerodynamic Design of Scramjet Facility Nozzles

Enrico Degregori[1] and Michele Ferlauto[1,a),b)]

[1]*Department of Mechanical and Aerospace Engineering, Politecnico di Torino, Turin, Italy*

[a)]Corresponding author: michele.ferlauto@polito.it
[b)]URL: http://staff.polito.it/michele.ferlauto

Abstract. An hybrid approach to optimization is applied to the aerodynamic design of a device that acts as a facility nozzle and inlet distortion generator for a direct-connect scramjet combustor test-rig. The solution of the design problem is sought by using different approaches of CFD analysis and optimization tools. The initial design is based on the solution of an inverse problem coupled to optimization by genetic algorithms. The result of the optimization is then tested by CFD simulations of the direct-connect facility and the shock reflection patterns are compared toat of the in-flight configuration. The verification of nozzle design and the assessment of the prescribed flow distortion are then carried out by numerical simulations of the flowfield of the whole direct-connect facility by an URANS solver. The proposed procedure is checked by designing a facility nozzle and distortion generator system for a scramjet model available in the open literature.

INTRODUCTION

The severe limitations imposed by the supersonic combustion in hypersonic air-breathing propulsion systems motivates the extensive Computational Fluid Dynamics (CFD) and experimental investigations on the aero-thermal coupling between hypersonic inlets, isolator and combustor [1, 2, 3, 4, 5, 6]. The CFD approach is somehow privileged and it is also used in designing the test facilities for the experimental investigations. Experimental testings on scramjet combustors are generally carried out in direct-connect scramjet (DCS) test-rigs, composed by a facility nozzle followed by the inlet, the isolator and the combustor. This configuration offers substantial advantages in terms of experimental costs, test duration, and experimental complexity with respect to the free-jet facility (e.g. inlet + isolator + combustor). Moreover, large-scale scramjets can only be investigated through direct-connect experiments [1]. In the traditional DCS test-rigs, a Facility Nozzle / Inlet Distortion Generator (FN-IDG) is placed before the inlet-isolator-combustor system that simulates the scramjet. This nozzle must generate the actual in-flight flow profile at a prescribed section of the scramjet inlet.

Aim of the present study is to assess an optimization procedure for the design of the facility nozzle system of the direct-connect test-rig and to analyze and assess the solution obtained in a multi-point design perspective. The key point is to establish an automatic design procedure that accurately defines a fixed geometry of the FN-IDG assembly, attached to the direct connect test-rig, which should replicate the unsteady evolution of the shock trains in the scramjet. This design and testing analysis is carried out by using both low order models and fully unsteady RANS simulations. The facility nozzle and the inlet distortion generator are considered as an integrated device and designed from scratch by using inverse methods coupled to an optimization techniques based on genetic algorithms. The geometry of scramjet facility proposed in Ref. [7] has been used as a reference. The CFD solution of the shock

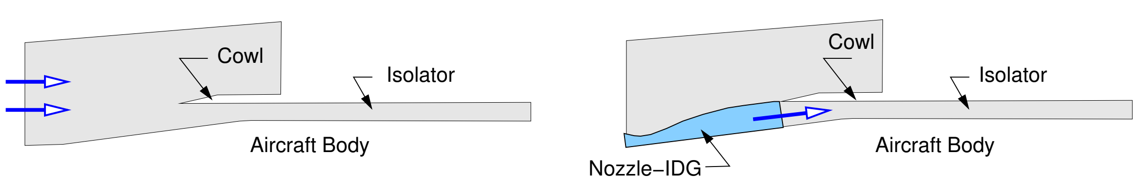
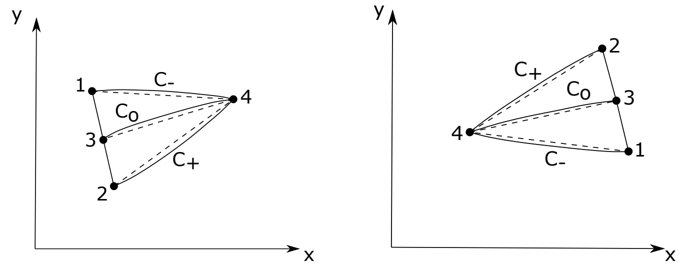


FIGURE 1. Sketch of the free-stream configuration and of the direct-connect facility model.

International Conference of Numerical Analysis and Applied Mathematics (ICNAAM 2017)
AIP Conf. Proc. 1978, 470114-1–470114-4; https://doi.org/10.1063/1.5048596
Published by AIP Publishing. 978-0-7354-1690-1/$30.00



(a) Forward (b) Backward

FIGURE 2. Method of Characteristics. Backward and forward projection of the wave signals.

structures and shock reflections generated by the designed FN-IDG attached to the direct-connect facility are compared to that generated in-flight. Finally the fully turbulent RANS solution of the direct-connect facility is studied.

MATHEMATICAL MODEL

The aerodynamic design problem is reformulated here as an inverse problem[7, 8] and coupled with an evolutionary optimizer based on Genetic Algorithms (GA)[9]. It is well known that GAs require a large number of iteration to converge. The solution of the inverse problem should rely therefore on a fast computational approach as the Method of Characteristics (MoC). In the following sections we outline the mathematical models used in the numerical procedure.

Method of Characteristics

The method of characteristics for steady supersonic flows is based on the Euler equations in quasi-linear form.

$$\begin{cases} \rho u_x + \rho v_y + u\rho_x + v\rho_y + \delta\rho v/y = 0 \\ \rho u u_x + \rho v u_y + p_x = 0 \\ \rho u v_x + \rho v v_y + p_y = 0 \\ u p_x + v p_y - a^2 u \rho_x - a^2 v \rho_y = 0 \end{cases} \tag{1}$$

where ρ, p, a are density, pressure and speed of sound, respectively; u, v are the flow velocity components. $\delta = 0$ holds for planar flow and $\delta = 1$ for axial-symmetric flow. The MoC transforms the PDE system (1) in the ODE system [10]:

$$\rho V dV + dp = 0, \qquad dp - a^2 d\rho = 0, \qquad \frac{\sqrt{M^2-1}}{\rho V^2} dp_\pm \pm d\theta_\pm + \delta\left[\frac{\sin\theta dx_\pm}{yM\cos(\theta \pm \alpha)}\right] = 0 \tag{2}$$

along the characteristic lines $(dy/dx)_o = \lambda_o = v/u$, $\quad (dy/dx)_\pm = \lambda_\pm = \tan(\theta \pm \alpha)$ where $\theta = \arctan(v/u)$ is the flow angle and $\alpha = \arctan(1/M)$. Dealing with rotational flows, three characteristics are passing through each point of the field: the streamline and two Mach lines. The developed numerical solution is based on a second-order accurate Euler predictor-corrector scheme and on a finite difference discretization [10]. We used GA optimization and the MoC for solving the inverse problem of deriving the facility nozzle shape from the knowledge of the pressure distribution on the nozzle upper-wall and of the flow solution along a certain curve in the flowfield, as explained in next sections.

Inverse Design using Forward MoC

A first method of solving the inverse problem is based on projecting the characteristics forward as shown in Figure 2a. In this case, the initial value line is the sonic line of the nozzle, which is approximated with a second order polynomial. The generic interior point is evaluated as the intersection of two Mach lines (C_+ and C_-) departing from the initial points 1 and 2 and the streamline (C_o) passing to the point 3. The pressure distribution, is represented by

$$p(x) = a_0 + a_1 x + a_2 x^2 + a_3 x^3 \tag{3}$$

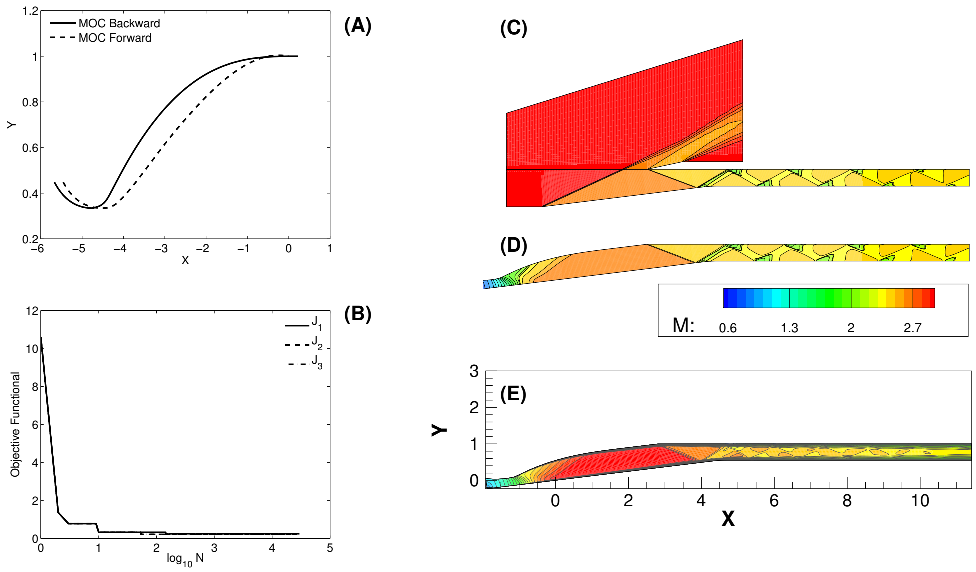


FIGURE 3. Design of the nozzle facility for $M_o = 3$. (A) Nozzle contour with forward and backward MoC design. (B) Objective functional optimization history. Inviscid computation of the in-flight condition (C) and of the direct-connect facility (D). RANS computation of the direct-connect facility (E). The Mach number contourmaps are shown.

and it is imposed as boundary condition on the upper wall. On the lower boundary the flow direction is imposed. The genetic algorithm code is used to optimize the shape of the nozzle, by taking the polynomial coefficients of eq. (3) as controls. The objective function to be maximized is defined as

$$J = \frac{\int_e ds}{\int_e [M - M_e(s)]^2 ds + \varepsilon} + \frac{\chi_1 \int_e ds}{\int_e [\theta - \theta_e(s)]^2 ds + \varepsilon} + \frac{\chi_2}{(h - h_e)^2 + \varepsilon} \tag{4}$$

in order match the prescribed distribution of Mach number $M_e(s)$, flow direction $\theta_e(s)$ and height h_e at the nozzle exit plane 'e' . The coefficients χ_1, χ_2 are functional weights and $\varepsilon = 10^{-12}$.

Inverse Design using Backward MoC

Another possible choice is to design the nozzle starting from the exit plane by projecting the characteristics backwards. The flow features are now prescribed straightly at the nozzle exit. The solution point is evaluated through the intersection of two backward-projected Mach lines and then the streamline is used to find point 3 (see Figure 2b). The boundary conditions for the upper and lower walls are the same of previous case. The solver stops when the sonic line of the nozzle is reached. The objective functional is

$$J = \frac{\int_i ds}{\int_i [M - M_i(s)]^2 ds + \varepsilon} \tag{5}$$

and it tries to find the pressure distribution at the upper-wall that better matches a prescribed location of the sonic line. This optimization is not strictly required but it is very helpful in the design of the subsonic part of the nozzle.

Flow Governing Equations

Outside the main loop of the GA-based optimization, we used different flow models. In-flight scramjet configurations was studied by using the Euler equations (1) written in divergence form and solved by a finite-volume method. The nozzle validation is carried out by a compressible Unsteady Reynolds Averaged Navier-Stokes equations (URANS) solver with Spalart-Allmaras (S-A) turbulence model [11]. The numerical solution is based on an OpenMP implementation of the shock-capturing Godunov method with Flux-Difference Splitting and second-order accurate Essentially Non-Oscillatory reconstruction scheme [12]. The URANS solver accuracy has been widely tested in many unsteady compressible flowfield involving stalled flows [13], moving grids [8], flow-instabilities [14].

NUMERICAL RESULTS

We consider the scramjet model of Yu et al. [7] as a reference. It simulates the flowpath of a simple configuration of inlet and isolator. The inlet is designed for a shock-on-lip Mach number of 4.0 over an inlet ramp inclined of 7 degrees and 314.5 mm long. A constant cross-sectional isolator (350 mm long and 30 mm high) is built beyond the inlet throat. The cowl is horizontal, with an height of 67 mm. The reference inlet condition of the test-case are: $P_o = 10$ kPa, $T_o = 300$ K and $M_0 = 3$. This simple geometry is able to generate an oblique shock and a shock-train that can represent the flow distortions and the typical flow characteristics in a scramjet.

The proposed optimization procedure is applied to the design of a device combining the features of both the facility nozzle and inlet distortion generator required to a direct-connect facility for the experimental testing of that scramjet model. We decided to select the cowl lip as reference section, that is, our designed device must replicate the flow conditions at that section. Firstly, the nozzle contour is derived by solving the inverse problem with both forward and backward MoC procedures, driven by the GA-optimization. The backward MoC results in faster converging design, since the penalization of the objective function is avoided. The flow pattern generated by the designed direct-connect facility nozzle at $M_0 = 3$ is checked against the corresponding flowfield of the scramjet computed at in-flight conditions. A good agreement in the generated shock patterns can be appreciated in Figure 3 . Since the nozzle design is based on inviscid models, the actual flow distortion at the reference section (i.e. the cowl inlet in our case) is then assessed by CFD simulations based on the compressible URANS solver. The computed Mach number flowfield is also depicted in Figure 3. As visible, the viscous effects and shock/ boundary-layer interactions have a strong (and beneficial) influence on the shock-train traveling towards the scramjet combustor. The flow exiting the facility nozzle is characterized by a viscous type velocity profile. In the core flow, the inviscid solution differs slightly from the viscous one in term of the reference Mach number. The error obviously increases close to the walls due to the different nature of boundary conditions in that region. Classic shape optimization of the final portion of the nozzle may match the correct boundary layer thickness at exit.

CONCLUSIONS

An hybrid approach to the optimal aerodynamic design of a facility nozzle of a direct-connect scramjet combustor test-rig has been illustrated. Numerical tools based on the method of characteristics coupled to a GA optimizer have been employed in the design of the facility nozzle by solving an inverse problem. The CFD simulations carried out on the candidate solution have shown that the solutions found are able to reproduce the correct shock reflection pattern when compared to the corresponding in-flight configuration. Improovements on the bondary layer characteristics at reference section can be obtained by including loss/boundary layer models in the inverse procedure with GA-optimization.

ACKNOWLEDGMENTS

Computational resources were provided by the Academic Computing project hpc@polito.it (http://www.hpc.polito.it).

REFERENCES

[1] S. Dunn, AIAA Paper 90-1309-CP, Orbital Debris Conference (1990).
[2] L. Riley, M. Hagenmaier, J. Donbar, and D. Gaitonde, AIAA SciTech Forum, AIAA-2017-0554 (2017).
[3] R. Boyce, A. Paull, and R. Stalker, J. Propuls. Power **16**, 718–720 (2000).
[4] C. Tam and et al., Journal of Propulsion and Power **29**, 1382–1390 (2013).
[5] M. Ferlauto, Proc. IMechE, Part A: Journal of Power and Energy **228**, 328–339 (2014).
[6] M. Hagenmaier, D. Eklund, and R. Milligan, AIAA Paper, 2011-233 (2011).
[7] K. Yu, J. Xu, L. Tang, and J. Mo, Aerospace Science and Technology **46**, p. 351359 (2015).
[8] M. Ferlauto, Inverse Problems in Science and Engineering **23**, 798–817 (2015).
[9] P. Charbonneau, NCAR Technical Note TN-450+IA (2002).
[10] M. Zucrow and J. Hoffman, *Gas Dynamics, vols. 1, 2* (John Wiley and Sons, New York, 1976).
[11] P. Spalart, F. Johnson, and S. Allmaras, 7^{th} Int. Conf. on Comput. Fluid Dyn., ICCFD7-1902 1–11 (2012).
[12] A. Harten, E. Engquist, and S. Osher, Journal of Computational Physics **71**, 231–303 (1987).
[13] M. Ferlauto and R. Marsilio, Adv. in Aircr. and Spacecr. Sci. **3**, 367–378 (2016).
[14] M. Ferlauto and S. Rosa Taddei, Proc. IMechE Part A: Journal of Power and Energy **229**, 359–366 (2015).