Investigation of nozzle stability for the first ovalization mode by numerical solution of the fluid–structure interaction problem

Richard Schwane[a,*], Yang Xia[a,b]

[a]ESA/ESTEC Postbus 299, 2200AG Noordwijk, The Netherlands
[b]AOES B.V. Leiden, The Netherlands

Abstract

Empirical and analytical methods have been used for the design of spacecraft engine components as the thermally and mechanically high-loaded expansion nozzles of present launch systems. Recent progress in high-performance computing facilities permits the numerical simulation of the unsteady flow fields and the interaction with structural dynamics. The objective of the present study is to investigate the degree to which such numerical solutions can reproduce observed phenomena that cannot be described by empirical or analytical models.

Keywords: Flow–structure interaction; Parallel algorithms; High-performance computing

1. Introduction

Technological developments in the launcher propulsion area are more and more focussed on reduction of launch costs. For a given launch system this means to increase the engine performance and the payload. Additionally, technologies have been implemented that allow for engine restart permitting multiple payloads with different destination orbits in one launch. Such performance improvements lead to reduced margins, e.g. for flow separation in performance-optimized expansion nozzles and for the interaction between flow, engine and nozzle structure.

Nozzle ovalization is considered as a critical load case for nozzle design, which is of importance for launch safety. This has predominantly two reasons. First, the low stiffness and high specific mass of the wall lead to an eigenfrequency of conventional nozzles for the first ovalization mode close to the frequencies where ambient flow fluctuations have maximal amplitudes. Second, in contrast to other structural modes, reinforcement of the structure to raise this first eigenfrequency implies heavy components that lead to reduced payloads. Additionally, testing of full-scale nozzles for excitation of ovalization modes is expensive and always incomplete, since the effect of ambient flow in the launcher base, see Fig. 1, and the effects of rarefied ambient conditions cannot be taken into account in ground tests. Computational Fluid Dynamic (CFD) methods are therefore an important tool for improved understanding of the flow physics that lead to ovalization instabilities or limit cycle oscillations that can result in material fatigue and consequently structural failure, normally causing the failure of the entire mission.

Existing nozzles have been designed based on engineering models describing the stability properties of nozzles [1,2]. All these models have three key simplifications of the flow problem in common:

- Assumption of flow in equilibrium to structure: only steady-state flow solutions are considered.
- Simplification of the flow model, e.g. to isentropic or non-rotational flow.
- An empirical model for the position of flow separation in the nozzle.

The present study aims at the investigation of the effect of the above assumptions on nozzle stability by application of a dynamical flow–structure model that avoids such simplifications. Present results demonstrate that nozzle destabilization can occur also in nozzles without flow separation inside a nozzle, in contradiction to the assumptions that underlie analytical modelling.

* Corresponding author. Tel.: +31 (71) 565 4370; Fax: +31 (71) 565 5421; E-mail: Richard.Schwane@esa.int

Computational Fluid and Solid Mechanics 2005
K.J. Bathe (Editor)

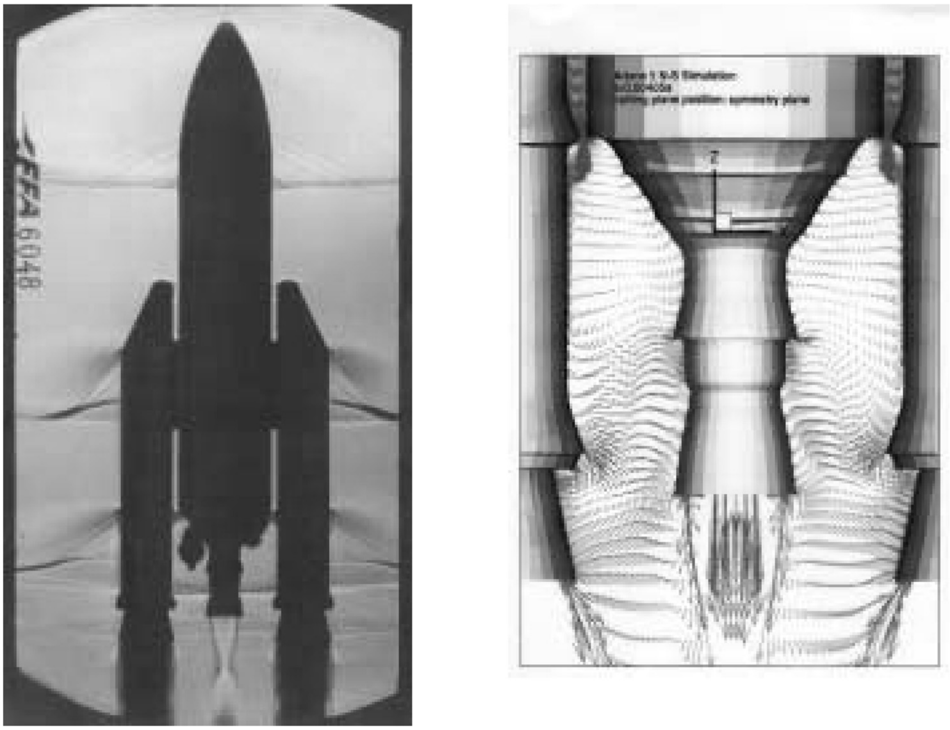

Fig. 1. Experimental schlieren picture and numerical snapshot of time-dependent flow field in the launcher base for transonic ambient flow, Ma = 0.85.

2. Validation and methodology

Results from previous investigations for nozzle deformation in the first bending mode, e.g. rigid nozzle rotation around the throat, have been used to validate the present methodology against analytical models from literature. The results have been published in [3]. The first bending mode was chosen since it permits the application of flow solvers that are time accurate on rigid moving grids; time-accurate mesh deformation is not required. A variety of such solvers is available, permitting thorough validation of the numerical method. In general, CFD data for bending reproduce well the predictions from analytical models. The method is based on the coupling of time-accurate Reynolds averaged Navier-Stokes (RANS) calculations to structural mechanics modelling. The paper will discuss in detail the numerical method used and its validation including the time accuracy of the algorithm for moving or deforming grids.

3. Application to nozzle ovalization

A number of observations based on dedicated experiments and flight data raised some doubts of the validity of the analytical models that have been used for nozzle design. The present study shall investigate to which extent the observed effects are caused by the inherent assumptions and simplifications of analytical models. Two parameters have to be selected for steady and forced harmonic ovalization, the amplitude of the deformation C and the distribution of the deformation past the nozzle axis, controlled by the exponent exp (see Fig. 5 for the definition of the parameters: $x = 0$: nozzle throat, $x = 1$ nozzle exit, $A(x)$ = radius deviation of deformed nozzle in the major axis from the radius of a circular nozzle). The selection of C and exp is then validated by the following steady-state procedure:

- Assume the amplitude and interpolation relation for the nozzle ovalization past the nozzle axis.
- Compute the pressure distribution for the deformed nozzle from a solution of the steady Navier-Stokes equations.
- Compute the structural deformation of an axisymmetric nozzle under this internal pressure distribution.
- Compare the resulting deformation with the original assumption.

If the resulting deformation agrees well with the assumption, the assumption is accepted as physically

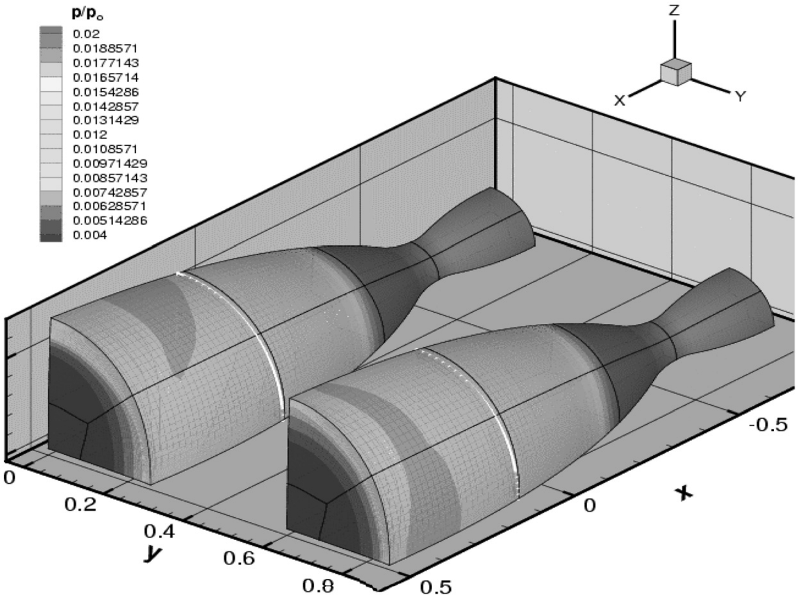

Fig. 2. Comparison of nozzle wall pressure: (left) progressive interpolation, exp > 1; (right) degressive interpolation, exp < 1.

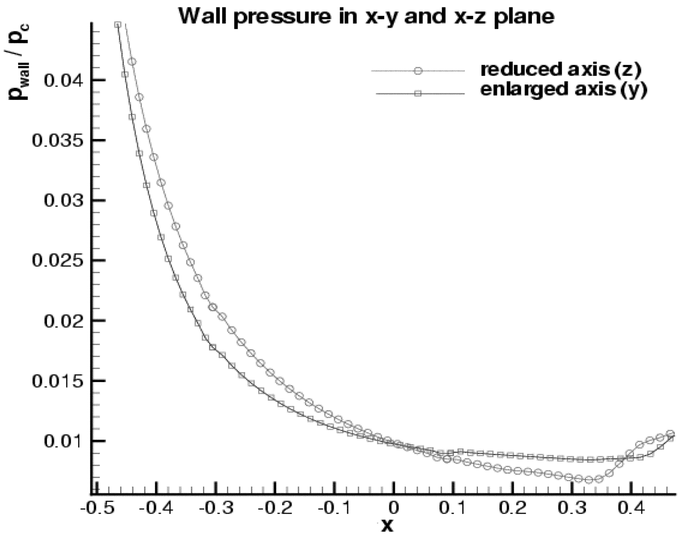

Fig. 3. Wall pressure in planes of major and minor axis of ovalized nozzle.

meaningful, since it describes a possible flow/structure coupled steady-state solution. In the present first study only metallic nozzles have been considered, with a wall thickness of 1 to 5 mm on a full-size ARIANE 5 flight scale. The described procedure shows the best match for a slightly over linear assumption of the ovalization amplitude, e.g. exp = 1–1.5. The wall thickness has been adapted to match the amplitude of the originally assumed and structurally computed deformation. The main conclusion from this study is that nozzle flow can have a destabilizing effect on ovalization even for fully attached flows. This destabilization that cannot be predicted by analytical models is particularly promoted for strongly degressive interpolation (exp < 1) of the ovalization amplitude along the nozzle axis.

4. Discussion of CFD results

Figures 2 to 5 illustrate the results of the present investigation for full flowing nozzles, or nozzles with

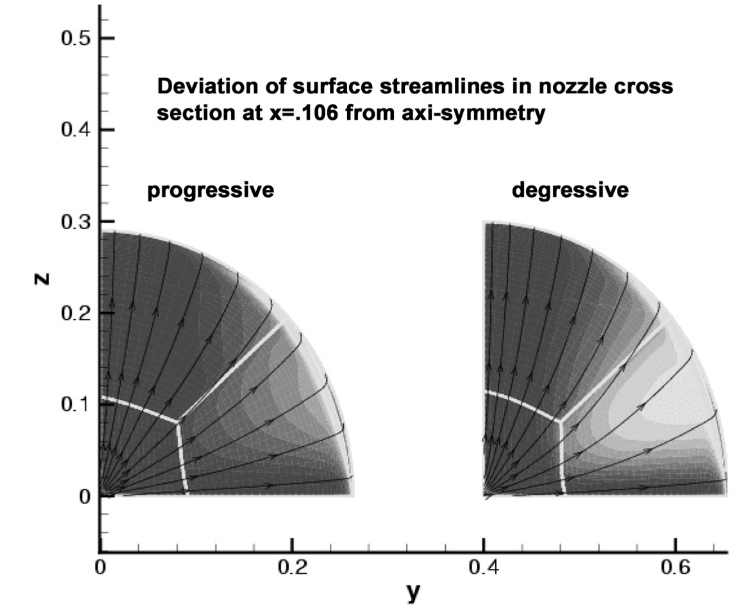

Fig. 4. Comparison of non-axisymmetric flow components.

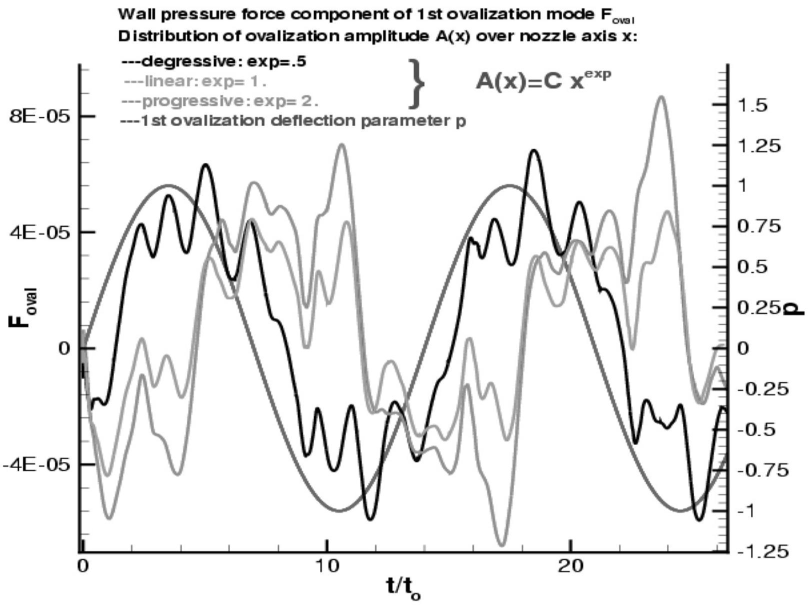

Fig. 5. Time history of force component for ovalization mode.

incipient separation. Figure 2 compares the wall pressure distribution of progressive interpolation (exp > 1, left) and degressive interpolation (exp < 1, right, see Fig. 5 for the definition). Shown are two snapshots from a time accurate computation taken at a normalized time $t/t_o = 2$ (in Figure 5). The comparison shows that only for the progressive case (left) the incipient shock supports the nozzle ovalization by further stretching the contour in the z-direction, whereas the compression shock is nearly axisymmetric and has no significant effect for the degressive case. In analytical models, as mentioned above, flow separation is considered as the main source of nozzle instability. The present example demonstrates, however, that flow separation has either a stabilizing, neutral or destabilizing effect on structural oscillations. Figure 3 shows the pressure distributions in the planes of the major and minor half axis with a cross-over at x = 0.05 for degressive interpolation. This cross-over leads to an effect of the separation shock on nozzle stability just reverse to the assumptions of the analytical models [1,2].

The statement is explained by the fact that a separation shock located downstream of the cross-over of the wall pressure distribution caused by secondary flow structures will show contradictory behaviour to the one assumed in Pekkari's original model [2]. The flow will separate earlier in the plane with the lower pressure, which is the major axis of every elliptic cross-section upstream and the minor axis downstream of the cross-over. Upstream separation in the minor axis, however, is stabilizing since it tries to restore the circular nozzle shape. This behaviour is explained in Fig. 4, where secondary flow in the nozzle is indicated by a contour plot of the deviation of the streamlines from the axisymmetric directions. All surface streamlines in cross sections run in a radial direction for an axisymmetric nozzle flow. For the progressive interpolation the deviation from axisymmetric flow is smaller than for the degressive case. The selected cross section is located at x = 0.106, as indicated by the white lines in Fig. 2. Finally, Fig. 5 shows the time history of the pressure force component in the ovalization direction for a progressive, linear and degressive case. It can be depicted that the degressive case deviates from the linear and progressive one, mainly by a much smaller phase shift between the ovalization parameter p (see Fig. 5) and the pressure force response. This results in a larger portion of one period where the ovalization force has the same sign as the ovalization parameter. For the same sign, the pressure force supports the ovalization and thus destabilizes the nozzle.

5. Conclusions

Due to the excitation of secondary, vortical flow structures, aerodynamic forces can also destabilize ovalized rigid and harmonically oscillating nozzles for fully attached flow. Key parameter of influence is the assumption for the oscillation amplitude versus the nozzle axis. Whether the coupled flow/structure system is finally stable or unstable depends on the relative magnitude of destabilizing aerodynamic force to the always stabilizing structural force. Based on the considered simplified thin-wall nozzle with 1 to 5 mm wall thickness the aerodynamic forces are on a level that permits nozzle instability.

References

[1] Frey M. Behandlung von Stroemungsproblemen in Raketenduesen bei Uberexpansion. PhD Thesis, TU Stuttgart, Germany, 2001.

[2] Pekkari L-O. Aero-elastic analysis of side load in supersonic nozzles with separated flow. In: Proc of the 30th AIAA/ASEM/SAE/ASEE Joint Propulsion Conference, Indianapolis, IN, 27–29 June, 1994, AIAA-94–3377.

[3] Schwane R, Xia Y. Time-accurate CFD predictions and data validation for side load generation by flow–structure coupling in over-expanded rocket nozzles. J Math Modelling Algorithms 2005 (to appear).