

Experimental and Numerical Study of Dual Bell Nozzle Flow

*C. Génin**, *R. Stark** and *O. Haidn**

**German Aerospace Center - DLR, Institute for Space Propulsion*

74139 Hardthausen, Germany

*K. Quering** and M. Frey***

***Astrium Space Transportation*

81663 Munich, Germany

Abstract

The dual bell is a nozzle concept for altitude adaption. The flow separates at the contour inflection in sea level mode in a mainly controlled and symmetrical way, reducing the side load generation and increasing the thrust. The transition to altitude mode is reached when the flow suddenly attaches to the extension for an improved altitude thrust. The conditions of this transition and its evolution are the key for the study of dual bell nozzles. For a better understanding of the flow behavior, a 2D subscale dual bell model has been designed and tested at the German Aerospace Center (DLR). The tests were divided into two campaigns and performed under cold and warm flow conditions. The evolution of the shock system at the inflection during the transition was observed using schlieren optics. The planar nozzle was tested under various conditions in pressure and temperature. Both test campaigns have been recalculated in cooperation with Astrium. Numerical and experimental results are presented in this paper.

1. Introduction

The main stage engine of the current launchers using parallel configuration (e.g. the European launcher Ariane 5) has to be ignited at ground for security reason. This imposes the limitation of the nozzle area ratio, reducing the potential overall performance. The nozzle works over-expanded at sea level and strongly under-expanded at altitude. Various concepts of altitude adaptive nozzles have been proposed in the literature to circumvent this limitation. The dual bell nozzle offers a very promising alternative to conventional nozzles [1].

A contour inflection links the base nozzle and the extension (Fig. 1) and forces the flow to a controlled and symmetrical separation in sea level mode. The side load generation is limited and the thrust increased. At a certain altitude, corresponding to a given nozzle pressure ratio (NPR), the transition to altitude mode takes place: the flow suddenly attaches to the extension wall down to the nozzle exit plane. The nozzle extension flows full and the whole area ratio is used, leading to a higher vacuum performance.

The concept of a contour inflection to control the flow separation was first proposed in 1949 by Foster and Cowles [2]. Many experimental and numerical studies have been conducted since then to prove the altitude adaptation qualities of the nozzle concept. In 1994, Horn and Fisher [3] investigated the influence of the extension contour geometry on the flow behaviour in a first experimental study. In Europe, the FESTIP research group realised feasibility and performance analysis [1, 4]. Various parametrical studies have also been realised to understand the flow phenomena and optimise the contour design [5, 6].

Cold flow subscale dual bell nozzle models have been tested intensively at the DLR [7]. Axisymmetrical nozzles were tested both under cold and warm flow conditions. Schlieren optics were used to investigate the flow evolution during the transition. For the present study, a 2D (planar) nozzle model has been chosen to permit the flow observation during the transition directly in the region of the contour inflection. The tests were conducted at the warm gas M11 test complex at DLR Lampoldshausen. In addition, a cold flow test campaign was conducted at the cold flow test facility P6.2, for transient investigation.

Representative test cases were then chosen and recalculated with a CFD tool. The comparison of the calculated results with the experimental data permits to validate the numerical method. Both experimental and numerical works were realized in the framework of the SFB – TR40 cooperation of the German Research Foundation.

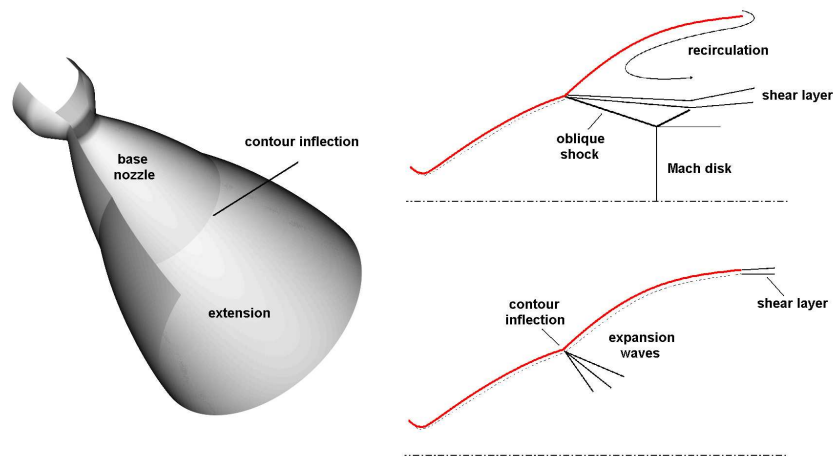


Figure 1: The dual bell nozzle and its two operating modes: sea level (top) and altitude mode (bottom).

2. Experimental setup

The experimental study was divided into two campaigns: a cold flow investigation at the P6.2 test facility and a warm flow study at the M11 test complex.

2.1 Cold flow test facility P6.2

The P6.2 test facility provides dry gaseous nitrogen obtained from 20 MPa high pressure vessels. The flow gets through a line system with an automatic valve, a filter, a pressure reducer and a mass flow meter. It reaches the settling chamber including a combination of grids and honey combs to reduce the turbulence intensity. The test specimen is mounted on the horizontal rig, downstream of the settling chamber and a section contraction, delivering a parallel flow.

The test facility supplies the nozzle with a feeding pressure up to 6 MPa and a mass flow up to 4.2 kg/s, under ambient temperature. The tests are conducted under ambient pressure conditions with up- and down ramping of the feeding gas pressure.

2.2 Test complex M11

The test complex M11 is divided into four test benches designed for the investigation of ramjets, scramjets and small supersonic nozzles. The test position M11.1 features hydrogen-oxygen burners heating pressurized air. The four-burner configuration in use for this test campaign yields total pressures up to 30 bar and temperatures up to 1500 K. Flow temperature and pressure can be adjusted by varying the hydrogen, oxygen and air mass flow rates. The hot flow was composed of air with about 5% water vapor for the test conditions presented in this study. The nozzle model was mounted on a horizontal rig (Fig 2). The tests were conducted under ambient conditions in pressure and temperature.

Figure 4 depicts a typical pressure and temperature evolution during a steady state test at test bench M11.1.

2.3 Nozzle model

The test specimen was designed for the hot flow investigation at the M11 facility. The model was a planar dual bell nozzle. The base nozzle was designed as a full-length ideal nozzle, to limit the 3D effects due to the side walls. The design Mach number is $M_D = 2.8$. The extension was defined as an isobar leaving the last point of the base nozzle (constant pressure extension). The design in-house program is based on the method of the characteristics. The theoretical transition nozzle pressure ratio is $NPR_{tr} = P_0/P_a = 25.8$. The geometrical parameters are summarized in table 1 and represented in Fig. 3.

The nozzle was constituted of two walls corresponding to the nozzle contour and two exchangeable parallel side plates. The nozzle depth was 45 mm and constant from the convergent nozzle part down to the exit plane. The wall

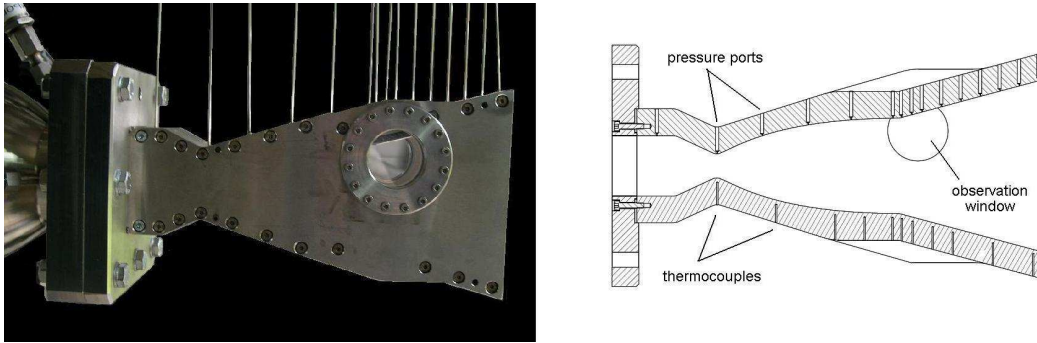


Figure 2: Planar dual bell nozzle model.

thickness for the side plates and the nozzle contours was 10 mm.

Throat radius	R_{th}	9 mm
Base length	L_b/R_{th}	15.1
Extension length	L_e/R_{th}	11.9
Area ratio	ϵ_b	3.9
	ϵ_e	7.1
Inflection angle	α	15°
Depth of the model	d	45 mm

Table 1: Geometry of the dual bell nozzle model.

2.4 Instrumentation

Wall pressure measurements were performed for the nozzle model. The pressure ports were placed along the nozzle wall, in the centre line of the upper nozzle contour. Small pipes welded in the wall connected the sensors to the 0.5 mm diameter orifices via small Teflon tubes (see Fig. 2). In addition, the pressures were measured in the side plates inside a port matrix placed in the observation window, permitting the comparison with the schlieren measurements.

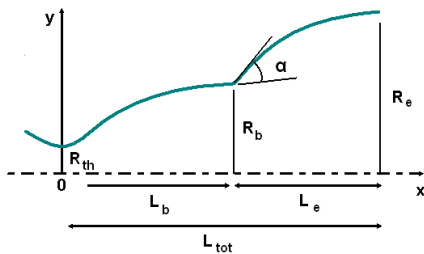


Figure 3: Geometrical parameters of a dual bell nozzle.

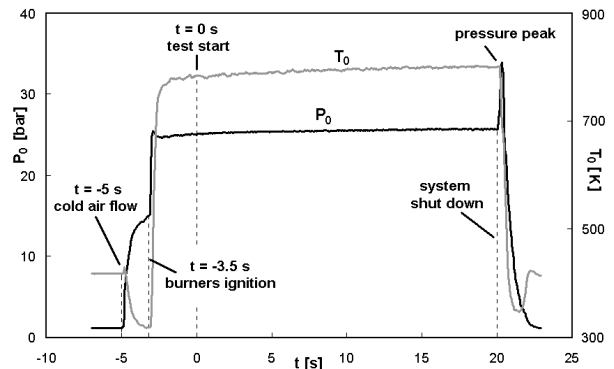


Figure 4: Typical total pressure and temperature variations during a warm flow test at M11.

A window of 45 mm diameter was located in the region of the contour inflection of the side plates. The flow evolution in this area has been observed using schlieren optics. Colored schlieren films were recorded for up- and down ramping of the NPR at the cold flow test bench. The density gradient variations of the flow in the inflection region visualize the separation shock system.

3. Numerical method

For the simulation of selected cold flow test cases a commercial 3D Navier-Stokes solver is used. The numerical model for the simulation of the planar dual bell nozzle and the boundary conditions are depicted in Fig. 5. The fluid is a mixture of dry nitrogen and air - nitrogen as working medium and air from the ambience. Due to symmetry reasons only one fourth of the nozzle is simulated and symmetry boundary conditions are used at the corresponding planes. At the inlet a total pressure in the range of 13 to 27 bar and a static temperature of 273 K were prescribed. The nozzle walls were set as adiabatic and assumed to be hydraulically smooth. Around the nozzle a control volume was constructed in order to take into account the interaction with the ambience. The grid resolving the depicted computational domain contained about 4 million nodes. At the control volume boundary in the direction of the nozzle inlet an inlet boundary condition with prescribed static pressure and temperature is used. In the direction of the nozzle outlet an opening boundary condition with prescribed static pressure and temperature is applied, which allows the outflow of air and nitrogen and the inflow of air. As turbulence model of the steady state RANS simulations the Menter Shear Stress Transport (SST) model was used.

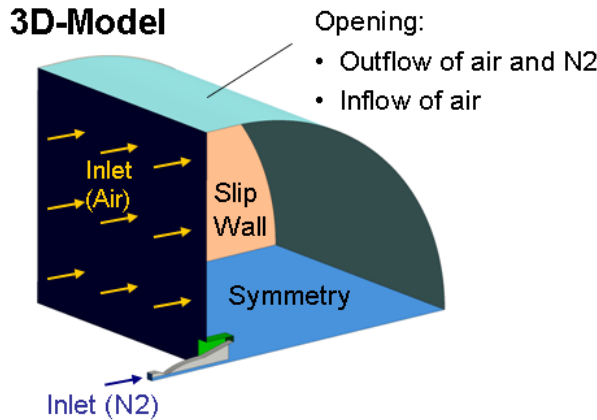


Figure 5: Boundary conditions for numerical simulation.

4. Results and discussion

4.1 Flow evolution in the nozzle

The wall pressure has been recorded along the nozzle contour for the various test configurations obtained at the test rig M11. Figure 6 depicts the wall pressure distribution for four values of NPR measured for the same total temperature $T_0 = 700 \text{ K}$.

For low NPR values (NPR = 20 and 24.2), the nozzle model is operating in sea level mode. The flow is attached in the base nozzle and separates at the contour inflection. When increasing the NPR value (24.7), the separation point starts moving into the extension nozzle. This phenomenon has been described as sneak transition [6].

However, when further increasing the NPR value, the separation point displacement continues progressively down to the nozzle exit. The expected abrupt transition to high altitude mode does not take place. This phenomenon indicates that the extension wall contour features a small negative pressure gradient.

The negative wall pressure gradient up to a $x/r_{th} = 20$, which is followed by a positive wall pressure gradient, is confirmed by a CFD simulation at altitude mode (see Fig. 7). Therefore, a distinct sneak transition takes place prior to the transition to nozzle full-flowing.

Test cases of different separation types were simulated with the commercial CFD solver for the cold flow conditions. At low nozzle pressure ratio (NPR = 13.5) flow separation takes place in the base nozzle, as depicted in Fig. 8. As the NPR is increased the separation moves to the contour inflection (NPR = 21.9) and is finally located in the region of the sneak transition (NPR = 27.2). The agreement between test and simulation is good for the two lower nozzle pressure ratios. For the highest nozzle pressure ratio the agreement is not as good as for the other ones. The flow separation occurs slightly further upstream than in the test, which is already known from simulations of conventional

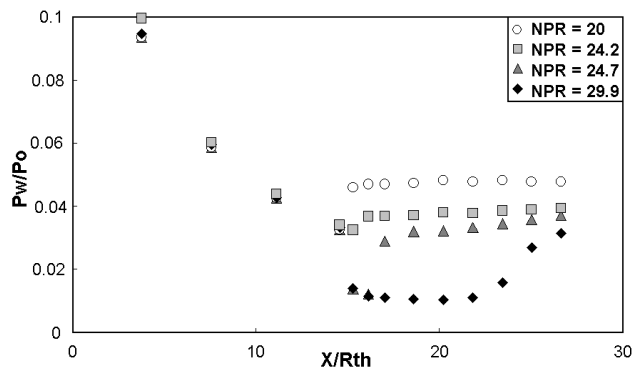


Figure 6: Wall pressure distribution for various NPR values.

nozzles. Additionally, the plateau pressure downstream of the flow separation is higher in the simulation than in the test.

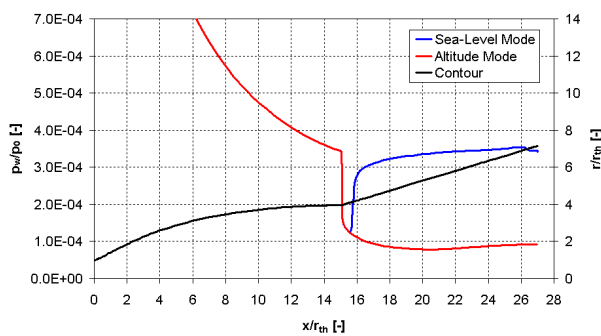


Figure 7: Wall pressure evolution in sea-level and vacuum mode.

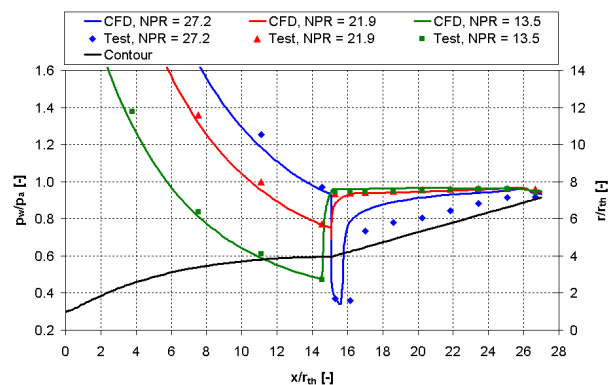


Figure 8: Comparison of wall pressure evolution in test and simulation.

4.2 Schlieren observation

The schlieren pictures were recorded with a rate of 50 frames per second. Figure 9 shows a series of four schlieren pictures taken during a cold flow transient test. The total pressure was progressively increased to observe the flow evolution during transition.

Picture a) and b) were taken for $NPR = 11.73$ and 16.38 . The separation shock system is located directly at the inflection: the nozzle flows in sea level mode. Picture c) corresponds to a NPR value of 25.02 . The separation point has started to move into the extension: the sneak transition takes place. In picture d) ($NPR = 27.37$), the separation point has moved further downstream into the extension.

The slight negative wall pressure gradient in the extension leads to a slower transition from sea level to altitude mode. This effect yields the possibility to observe in detail the evolution of the separation shock system. The position and the shape of the separation shock have been measured from the schlieren pictures using an in-house tracing program and Matlab. Figure 10 illustrates the position of the separation shock in the observation window for various NPR values, from sea level mode to almost altitude mode.

The angle between the separation shock and the extension wall has been recorded while varying the NPR value. At low NPR , the flow separates in the base nozzle. The angle decreases progressively while the jet opens. In sea level mode, the separation point is fixed at the contour inflection and the jet continues widening. Once the separation point starts moving into the extension, the angle remains almost constant. A similar effect can be seen when lowering the NPR , like during the retransition. The evolution of the jet angle determines the shape of the recirculation area in the

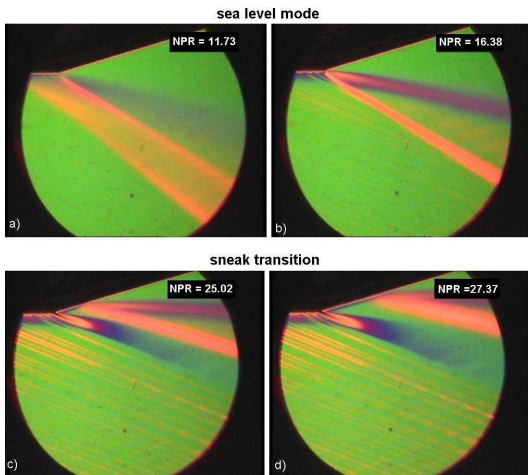


Figure 9: Observation of the flow in the extension region with color schlieren.

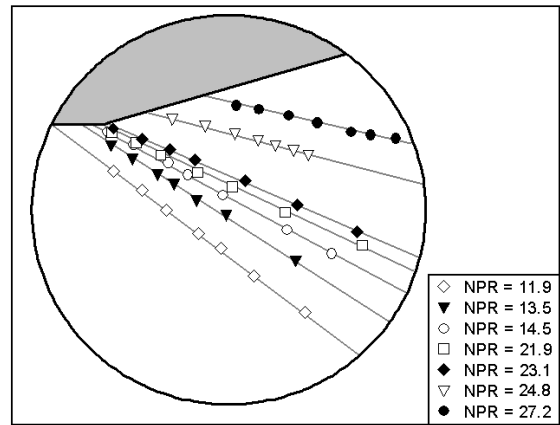


Figure 10: Evolution of the separation angle during NPR up- and down-ramping.

extension in sea level mode, and hence, the pressure downstream of the separation point.

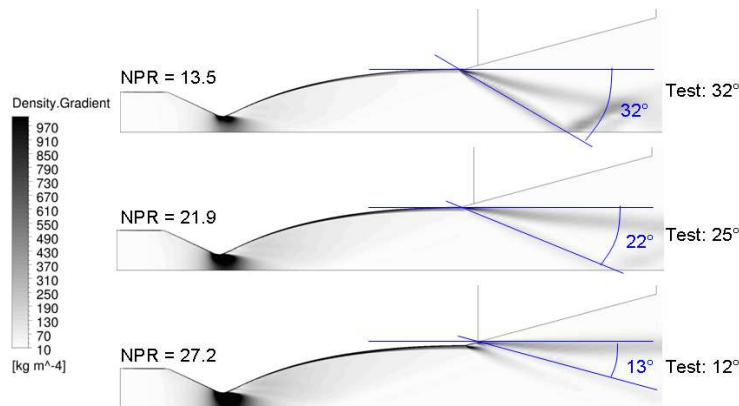


Figure 11: Evaluation of shock angle with density gradient contour plots.

A second angle is defined between the horizontal and the separation shock. These measured angles were then compared to the CFD simulations, where the angles were determined by contour plots of the density gradient (Fig. 11). Although the evaluation of the angle is not very precise, it gives nevertheless a hint for the modeling capability of the shock. In all investigated test cases the angles of CFD and test were close together, i.e. no higher deviation was observed for $NPR = 27.2$, where slight differences in the wall pressure were present.

4.3 3D effects

The test facility M11.1 limits the maximal depth for a 2D nozzle to $d = 45$ mm. The depth over width ratio, $d/R_{th} = 2.5$, is rather low at the nozzle throat and becomes very small at the nozzle exit $d/R_{th} = 0.35$. This leads to a 3 dimensional behaviour of the flow in the nozzle. One effect can be seen in Fig. 12, a schlieren picture of the flow for a low NPR value.

The separation shock appears to be very thick, from a position in the base nozzle down to the inflection. The graphic in Fig12 (right) is the wall pressure distribution corresponding to the schlieren picture. The flow separates in the centre line of the nozzle contour at the most up-stream position. On the side walls, the flow is already attached at the inflection, forming a “U”-flow pattern.

As can be seen from the CFD simulation, the separation line ($\tau_{wall} = 0$) is not constant over the nozzle depth

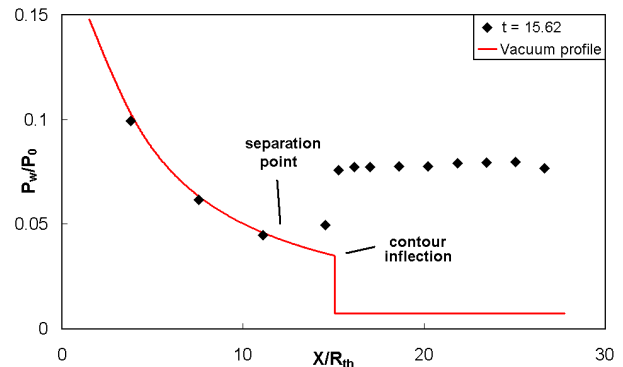
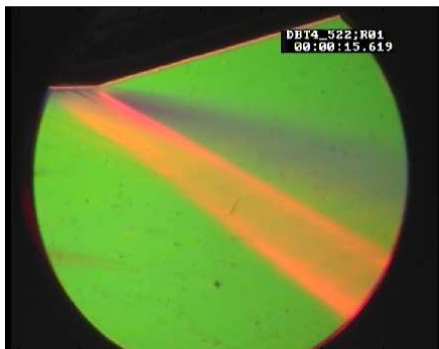


Figure 12: Evolution of the separation angle during NPR up- and down-ramping.

(Figure 13). In the corner between the side plate and the contoured wall, flow separation takes place directly at the contour inflection, while in the nozzle symmetry plane the flow is already in the sneak transition mode. —

In figure 13 (left), where the separation line ($\tau_{wall} = 0$) is depicted, a similar behaviour can be observed in the numerical simulation at low NPR: The flow separates at the centerline upstream of the contour inflection, while it is already attached to the contour inflection in the region close to the side plate. However, for higher NPR the situation changes (Fig. 13, right). In the corner between the side plate and the contoured wall, flow separation takes place directly at the contour inflection, while in the nozzle symmetry plane the flow is already in the sneak transition mode.

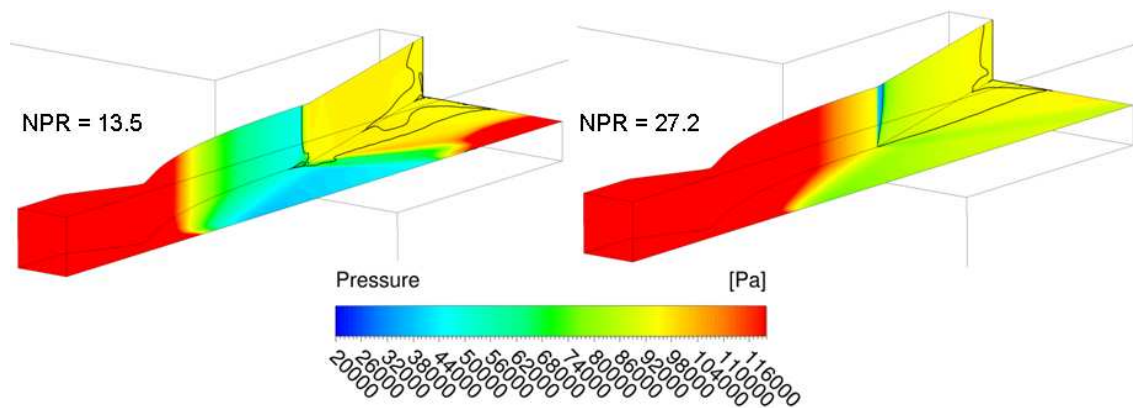


Figure 13: 3-dimensional shape of separation line ($\tau_{wall} = 0$).

5. Conclusion

A planar dual bell nozzle model has been tested under various test conditions in cold and warm flow. The observation of the shock system in the region of the contour inflection gives information on the position and the shape of the separation front. In sea level mode, the experimental and numerical results are in good agreement. By higher NPR values, the calculated separation position is located further up-stream than measured in the experiments. Despite these small differences with the experiments, the numerical method yields a good simulation from the flow behavior and helps to improve the understanding of the physics of the flow.

Acknowledgments

Financial support has been provided by the German Research Foundation (Deutsche Forschungsgemeinschaft – DFG) in the framework of the Sonderforschungsbereich Transregio 40.

References

- [1] H. Immich and M. Caporicci. 1996. FESTIP Technology Developments in Liquid Rocket Propulsion for Reusable Launch Vehicles. In: *Proceedings of 32nd AIAA Joint Propulsion Conference, 1-3 July 1996, Lake Buena Vista, FL*, AIAA-1996-3113.
- [2] C. Foster and F. Cowles. 1949. Experimental study of gas-flow separation in overexpanded exhaust nozzles for rocket motors. *Progress Report*, No. 4-103, Jet Propulsion Laboratory.
- [3] M. Horn and S. Fisher. 1994. Dual-bell altitude compensating nozzles. *Rocketdyne Division*, NASA-CR-194719.
- [4] G. Hagemann, M. Frey and D. Manski. 1997. A critical assessment of dual bell nozzles. In: *Proceedings of the 33rd Joint Propulsion Conference, Seattle, WA*, AIAA-1997-3299.
- [5] E. Martelli, F. Nasuti and M. Onofri. 2007. Numerical Parametric Analysis of Dual-Bell Nozzle Flows, *AIAA Journal*, No. 45, pp 640-650.
- [6] C. Génin. 2010. Experimental Study of Flow Behaviour and Thermal Loads in Dual Bell Nozzles. Ph.D. thesis, Université de Valenciennes, France. ISBN 978-3-8322-9230-0.
- [7] C. Nürnberger-Génin, R. Stark, H. Ciezki and O. Haidn. 2009. Experimental study of transition behaviour in high adaptive dual bell nozzles. SFB/TRR 40 - Annual Report 2009.
- [8] M. Frey, A. Preuss, S. Girard, Th. Alziary de Roquefort, Ph. Reijasse, R. Stark, K. Hannemann, R. Schwane, D. Perigo, L. Boccaletto and H. Lambaré. 2005. Joint Effort Towards Advanced Rocket Thrust Chamber Technology. In: *Proceeding of 6th Internat. Symposium on Launcher Technologies, Munich, Germany*.
- [9] G. Hagemann, A. Preuss, F. Grauer, M. Frey, J. Kretschmer, R. Ryden, K. Jensen, R. Stark and D. Zerjeski. 2004. Flow Separation and Heat Transfer in High Area Ratio Nozzles. In: *Proceedings of 40th AIAA Joint Propulsion Conference, 11-14 July 2004, Fort Lauderdale, FL*, AIAA-2004-3684.
- [10] M. Frey, R. Stark, H. Ciezki, F. Quessard and W. Kwan. 2000. Subscale nozzle testing at the P6.2 test stand. In: *Proceedings of 36th AIAA Joint Propulsion Conference, Huntsville, AL*, AIAA-2000-3777.