Operation Mode Transition of Film-Cooled Dual-Bell Nozzles

By D. Schneider†, K. Kostyrkin‡, R. Stark†, C. Génin† and M. Oschwald†

†Institute of Space Propulsion, German Aerospace Center (DLR)
Im langen Grund, Hardthausen, D-74239, Germany
‡RWTH Aachen University, Templergraben 55, Aachen

A numerical study is conducted to investigate the impact of a film-cooled dual-bell nozzle extension on its operation mode transition behavior. Therefore, unsteady Reynolds-averaged Navier-Stokes simulations of the transition process between sea level and altitude mode are carried out. The investigated dual-bell nozzle model is fed with hot gas by a combustion chamber using liquid oxygen as oxidizer and gaseous hydrogen as fuel. Upstream of the dual-bell nozzle contour inflection gaseous hydrogen is injected as cooling fluid for the nozzle extension wall. The numerical studies yield a clear impact of the cooling fluid mass flow rate on the transition nozzle pressure ratio of the dual-bell nozzle. The increase of the cooling fluid mass flow rate leads to a shift of the dual-bell transition nozzle pressure ratio to lower values. Furthermore, the impact of the combustion chamber mixture ratio on the dual-bell operation mode transition is investigated. A clear shift to lower transition nozzle pressure ratio values due to higher propellant mixture ratios can be observed. A combination of the two effects is introduced for an active control of the dual-bell operation mode transition.

1. Introduction

The dual-bell nozzle [1–4] (DBN) is an altitude adapting nozzle concept. It combines the advantage of a nozzle with small expansion ratio under sea level conditions and a nozzle with high area ratio under high altitude conditions. The main stage engine of modern parallel staged launchers like the European heavy launcher Ariane 5 is ignited prior to the booster stage because of safety reasons. This means the main stage engine has to operate over a wide range of altitudes from sea level up to almost vacuum conditions. Due to potential flow separation under high ambient pressure conditions, the expansion ratio of conventional nozzles is limited. The consequence of the expansion ratio limitation is reduced performance under high altitude conditions. The dual-bell nozzle consists of a base nozzle and a nozzle extension linked by an abrupt change in wall angle. This contour inflection forces the flow to a controlled and symmetrical separation under sea level conditions. During ascent of the launcher the ambient pressure decreases. After reaching a certain nozzle pressure ratio (ratio between combustion chamber total pressure and ambient pressure), the flow separation position moves abruptly to the exit area of the dual-bell nozzle extension, and a higher expansion ratio is available for a wide range of the launch trajectory. This increase of engine performance yields a potential payload gain [5]. Figure 1 shows a principle sketch of the supersonic flow field of a
A convergent-divergent nozzle with a contour inflection in the supersonic part to control the flow separation was first proposed by Chalom [6] and Foster et al. [7]. In 1965 Fischer [8] published the idea of a one step altitude adaptation with resulting engine performance increase using a dual-bell nozzle. First cold-flow investigations were conducted by Horn and Fisher [9] in the 1990s. The study focused on the verification of the two operation modes and the investigation of the flow behavior. Horn and Fisher calculated a payload gain of 12.1% for a one stage launcher with three space shuttle main engines and dual-bell nozzle application. Numerical simulations of the Horn and Fisher experiments were conducted by Goel et al. [10] in 1995. In the end of the 1990s the growing interest in single stage to orbit applications led to several dual-bell nozzle studies in Europe and Japan. Experiments using cold and warm air, numerical investigations and analytical studies were conducted in Germany by Haidinger et al. [11] and Manski et al. [12] in 1998. In the end of the millennium Kumakawa [13] and Kusaka [14] conducted dual-bell hot-flow experiments with hypergolic propellants.

Current studies focus on the flow behavior of dual-bell nozzles using inert cold gas as the fluid. The experimental investigations of Dumnov et al. [15], Kimura et al. [16] and Génin et al. [17] analyzed the impact of different geometrical parameters on the performance of liquid rocket engines with dual-bell nozzles. The studies of Tomita et al. [18], Reijasse et al. [19], Verma et al. [20–22] and Génin et al. [23, 24] investigated the flow behavior during the transition from sea level to altitude mode. At this time the definition of sneak transition was introduced and studied by Génin et al. [4]. Furthermore, Stark et al. [25], Génin et al. [17, 23] and Verma et al. [26] investigated the hysteresis effect of the dual-bell nozzle. Additional thermal loads due to the contour inflection were studied by Génin et al. [24] using inert hot air flow. The side loads occurring during operation mode transition of a dual-bell nozzle were experimentally studied by Génin et al. [27] in 2011. Nasuti et al. [28, 29] investigated alternative dual-bell design methods to the method of characteristics. The concept study of Jones et al. [30] in 2014...
verified the feasibility of a dual-bell nozzle experiment below a F-15 NASA Jet, to provide realistic flight conditions during test.

First transient numerical simulations of the dual-bell operation mode transition in the case of a constant pressure extension and inert cold gas flow were conducted by Wong et al. [31], Nasuti et al. [32] and Karl et al. [33] in the beginning of the last decade. In these studies the feasibility of the numerical prediction of the dual-bell operation modes and the transition between them was proven. An improved dual-bell transition behavior was shown by Génin et al. [34] for a nozzle with positive pressure gradient along the inner extension wall. Perigo et al. [35] investigated the impact of combustion chamber and ambient pressure fluctuations on the dual-bell transition behavior. They postulated a minimum dual-bell hysteresis of 20 % to avoid a flip-flop behavior. The study of Martelli et al. [36] shows a dual-bell inflection region in the same order as the throat diameter. Furthermore, the inflection region decreases with increasing Reynolds number and a relatively small feeding pressure gradient has no influence on the transition duration. Nasuti et al. [29] numerically confirmed these findings. An experimental validation of these results is still pending. The impact of the dual-bell extension design on the transition behavior was investigated by Martelli et al. [37] in 2005. The simulations have shown that for a dual-bell nozzle with constant pressure extension flow separation can occur between the contour inflection and the nozzle exit plane. A change of the extension design to a positive pressure gradient contour leads to an improved transition behavior. These results have been confirmed by the study of Génin et al. [38] in 2013. Schneider et al. [39] published a detailed work on the numerical prediction of the transition and hysteresis behavior of cold-flow dual-bell nozzles. In this study the influence of the turbulence model and the feeding pressure gradient on the dual-bell transition and retransition was investigated.

The dual-bell transition behavior under LOX/H₂ hot-flow conditions was numerically studied by Martelli et al. [40] in 2007. No validation data have been available for these numerical simulations. Within a cooperation between DLR and JAXA, Génin et al. [41] and Takahashi et al. [42] investigated experimentally the dual-bell transition behavior under LOX/CH₄ hot-flow conditions. A clear impact of the combustion chamber mixture ratio on the transition nozzle pressure ratio was observed. Based on this experimental work Schneider et al. [43, 44] developed a validated numerical method to predict the dual-bell transition and hysteresis behavior under LOX/CH₄ hot-flow conditions. The clear impact of the combustion chamber mixture ratio on the transition nozzle pressure ratio was reproduced by these numerical simulations.

Many main stage engines like the Vulcain 2 engine of the European heavy launcher Ariane 5 use a combination of regenerative cooling and film cooling, due to the extremely high thermal loads at the engine walls. In case of the film cooling a secondary gas is injected to the inner engine wall to thermally insulate the wall from the hot gas flow [45]. The impact and effectiveness of a cooling film on the wall heat flux in a liquid rocket thrust chamber was investigated by Arnold et al. [46] under real application conditions. This study yields a linear relationship between the cooling film blowing rate and the cooling efficiency.

Proshchanka et al. [47,48] first studied the injection of a secondary gas inside a dual-bell nozzle to control the operation mode transition. The experimental and numerical studies yielded that the axisymmetric film coolant injection behaves in the same manner as wall normal film coolant injection. Martelli et al. [37] studied the impact of film-cooling on the dual-bell inflection region. The authors identified the boundary layer thickness at
the end of the nozzle base, the Prandtl-Meyer expansion at the contour inflection and the wall pressure gradient of the nozzle extension as key parameters for the shape of the inflection region. Film-cooling is identified as particularly effective if the film is injected in the nozzle base. For an ascending launcher configuration the flow separation position can spend up to 10 s in the inflection region during transition. Assuming a averaged launcher velocity of \(317.5 \text{m/s}\) during the sneak transition this corresponds to altitudes between \(5632\text{m}\) and \(8290\text{m}\). Furthermore, Martelli et al. [49] investigated the effect of secondary gas injection on the transition behavior of dual-bell nozzles and the cooling efficiency for a Vulcain 2 like \(\text{O}_2/\text{H}_2\) configuration. Good cooling efficiency was achieved for high supersonic Mach number with reduced injection temperature. A relatively large inflection region up to 50 % of the extension length was observed for a film-cooled dual-bell nozzle. Another cold-flow study of Martelli et al. [50] yielded a decreased operation mode transition duration due to film coolant injection.

**Experimental Setup**

The present numerical investigations are the preliminary design work for hot-flow tests conducted in the framework of the Sonderforschungsbereich Transregio 40 of the German Research Foundation (DFG). The tests will be carried out in 2018 at the European research and technology test facility P8 of the German Aerospace Center (DLR) in Lampoldshausen. The primary objective of the hot-flow experiments is the investigation of the film-cooling impact on the dual-bell operation mode transition behavior. Figure 2 illustrates the thrust chamber assembly (TCA) for the experimental dual-bell nozzle study.

The combustion chamber (CC) used for the hot-flow tests is the BK PTE [51], which was already operated by DLR in previous campaigns. The water-cooled cylindrical copper part of the combustion chamber has an inner diameter of \(d_{cc} = 50\text{ mm}\). BK PTE features an injection head with 15 coaxial injector elements. The propellant combination used for the test campaign is liquid oxygen and gaseous hydrogen (LOX/\(\text{H}_2\)).

The cylindrical part is followed by the water cooled nozzle throat segment (NT), which has a throat diameter of \(d_{th} = 33\text{ mm}\). The throat segment ends at a area ratio of \(A_{IF1}/A_{th} = 5\) and is followed by the dual-bell nozzle part. This part is divided into dual-bell base nozzle (DB Base) and nozzle extension (DB Extension). The section downstream of the nozzle throat is designed as a truncated ideal contour, to avoid the appearance of an inner shock [52] inside the DBN flow field. The base nozzle has a design Mach number of \(M_D = 4.5\). The DBN base nozzle section features the film cooling injection part and is regeneratively cooled using \(\text{H}_2\) flowing through multiple
### Table 1. Geometrical parameters of the investigated dual-bell nozzle model.

<table>
<thead>
<tr>
<th>Name</th>
<th>Symbol</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Combustion chamber diameter</td>
<td>$d_{cc}$</td>
<td>50 mm</td>
</tr>
<tr>
<td>Throat diameter</td>
<td>$d_{th}$</td>
<td>33 mm</td>
</tr>
<tr>
<td>Design combustion pressure</td>
<td>$P_{cc}$</td>
<td>10 MPa</td>
</tr>
<tr>
<td>Oxidizer / Fuel</td>
<td>LOX</td>
<td>6.0</td>
</tr>
<tr>
<td>Mass ratio Oxidizer / Fuel</td>
<td>o/f</td>
<td>6.0</td>
</tr>
<tr>
<td>Vacuum thrust</td>
<td>$F_{vac}$</td>
<td>16 kN</td>
</tr>
<tr>
<td>Fuel injection</td>
<td>Coaxial</td>
<td>15</td>
</tr>
<tr>
<td>Base nozzle design Mach number</td>
<td>$M_{D}$</td>
<td>4.5</td>
</tr>
<tr>
<td>Total nozzle length</td>
<td>$L_{D}/R_{th}$</td>
<td>14.8</td>
</tr>
<tr>
<td>Interface TC/base nozzle</td>
<td>$A_{IP1}/A_{th}$</td>
<td>5.0</td>
</tr>
<tr>
<td>Film injection</td>
<td>$A_{film}/A_{th}$</td>
<td>9.5</td>
</tr>
<tr>
<td>Interface base/extension nozzle</td>
<td>$A_{IP2}/A_{th}$</td>
<td>14.8</td>
</tr>
<tr>
<td>Contour inflection</td>
<td>$A_{inf}/A_{th}$</td>
<td>17.7</td>
</tr>
<tr>
<td>Contour inflection angle</td>
<td>$\alpha_{inf}$</td>
<td>14.7°</td>
</tr>
<tr>
<td>Nozzle exit area ratio</td>
<td>$A_e/A_{th}$</td>
<td>53.0</td>
</tr>
<tr>
<td>Nozzle exit angle</td>
<td>$\alpha_e$</td>
<td>13.0°</td>
</tr>
</tbody>
</table>

Rectangular cooling channels. Before the GH$_2$ is injected parallel to the nozzle wall at $A_{film}/A_{th} = 9.5$, it is accelerated by convergent/divergent nozzles to supersonic velocity.

The interface between DBN base and extension is located upstream of the DBN contour inflection at $A_{IP2}/A_{th} = 14.8$. This opens the possibility to use different DBN extension in future studies. The DBN extension is designed using a DLR in-house tool based on the method of characteristics [23]. A positive pressure gradient (PP) extension is applied for the present investigation. Former studies of the authors [38, 39] yielded a fast and well defined dual-bell transition to altitude operation mode for a PP extension. The contour inflection is located at an area ratio of $A_{inf}/A_{th} = 17.7$ and features a contour inflection angle of $\alpha_{inf} = 14.7^\circ$. The nozzle exit area ratio is $A_{e}/A_{th} = 53.0$ with an exit angle of $\alpha_e = 13.0^\circ$. The film cooling section and the DBN extension are produced using additive layer manufacturing. All details of the combustion chamber and the dual-bell nozzle model are given in Table 1. An extensive overview of the design process of the dual-bell nozzle demonstrator engine is given by Stark et al. [53].

### 2. Numerical Investigation

#### 2.1. Numerical Method

##### 2.1.1. The DLR-TAU Code

The numerical study is performed with the second order finite-volume DLR-Navier-Stokes flow solver TAU [54], which has been validated for a wide range of steady and unsteady sub-, trans- and hypersonic flows [55]. The integral form of the Navier-Stokes equations for a mixture of chemically reacting ideal gases is read:

$$\rho \frac{\partial}{\partial t} \iiint_V \vec{U} \, dV + \iiint_A \vec{F}_{inv} \, n \, dA = \iiint_A \vec{F}_{visc} \, n \, dA + \iiint_V \vec{S} \, dV$$

(2.1)
The vector of conservative variables can be written as:

$$\vec{U} = \begin{pmatrix} \rho_s \\ \rho u^T \\ \rho E \end{pmatrix}$$

(2.2)

The inviscid Euler fluxes are:

$$\vec{F}_{\text{inv}} = \begin{pmatrix} \rho_s \vec{u}^T \\ \rho \vec{u} \vec{u}^T \\ \rho H \vec{u} \end{pmatrix} + \begin{pmatrix} 0 \\ \rho \vec{I} \\ 0 \end{pmatrix}$$

(2.3)

and the viscous Navier–Stokes fluxes read:

$$\vec{F}_{\text{visc}} = \begin{pmatrix} \rho D \nabla T \rho_s \\ \rho D \sum_s h_s \nabla T \rho_s + (\vec{p} \vec{u})^T \end{pmatrix}$$

(2.4)

The Boussinesq approximation was used to model the viscous stress tensor $\vec{P}$:

$$\vec{P} = \mu \left( \nabla \vec{u}^T + (\nabla \vec{u}^T)^T \right) - \frac{2}{3} \mu \left( \nabla^T \vec{u} \right) \vec{I}$$

(2.5)

The source from the chemical reactions $\omega_{s}$ and the volumetric heat source $\dot{q}$ are included in the source vector:

$$\vec{S} = \begin{pmatrix} \omega_s \\ \dot{q} \end{pmatrix}$$

(2.6)

The ideal gas equation closes the system:

$$p = \sum_s \rho_s R_s T$$

(2.7)

The laminar viscosity of each individual species was calculated applying Blottner curve fits [56]:

$$\mu_s = \frac{1}{\left( \frac{N_s}{m^2} \right)} \exp \left( C_s \right) T^{(A_{s,\text{ln}(T)}+B_s)}$$

(2.8)

The Blottner coefficients $A_s$, $B_s$, and $C_s$ were obtained from the NASA CEA software and database of Gordon and McBride [57]. The overall laminar viscosity for the gas mixture was calculated applying Wilke’s mixing rule [58]:

$$\mu = \sum_s \sum_m n_{s,m} \mu_s \Phi_{s,m}$$

(2.9)

where:

$$\Phi = \frac{1}{\sqrt{8}} \left( 1 + \frac{M_s}{M_m} \right)^{-\frac{1}{2}} \left[ 1 + \left( \frac{\mu_s}{\mu_m} \right)^{\frac{1}{2}} \left( \frac{M_m}{M_s} \right)^{\frac{1}{4}} \right]^2$$

(2.10)

An additional transport coefficient is the thermal conductivity $k$, which was determined for the individual species by using the modification of the Eucken correction by Hirschfelder [59]:

$$k_s = \mu_s \left( \frac{5}{2} (c_v)^{tr}_s + \frac{(c_v)_s - (c_v)^{tr}_s}{Sc} \right) \quad \text{with} \quad (c_v)^{tr}_s = \frac{3}{2} \frac{R_s}{M_s}$$

(2.11)
Here, the Schmidt number was 0.7 for all species. The overall heat conductivity of the gas mixture was calculated following the mixture rule of Zipperer and Herning [60]:

\[
k = \sum_s \frac{n_s k_s}{\sum_m n_s \sqrt{M_m/M_s}}
\]

(2.12)

The diffusion coefficient \( D \) in Fick’s law was computed:

\[
D = \frac{\mu}{\rho S c}
\]

(2.13)

Assuming a reacting mixture of thermally perfect gases the specific heat \( c_p \), the specific enthalpy \( h \) and the specific entropy \( s \) of each individual species were tabulated between 200 K and 6000 K using curve fits following McBride et al. [61]:

\[
\frac{c_{p,s}}{R} = a_{1,s} + a_{2,s}T_s + a_{3,s}T_s^2 + a_{4,s}T_s^3 + a_{5,s}T_s^4
\]

(2.14)

\[
\frac{h_s}{RT_s} = a_{1,s} + a_{2,s}T_s + \frac{a_{3,s}T_s}{2} + \frac{a_{4,s}}{3}T_s^2 + \frac{a_{5,s}}{4}T_s^3 + \frac{a_{6,s}}{5}T_s^4 + a_{7,s}
\]

(2.15)

\[
\frac{s_s}{R} = a_{1,s}lnT_s + a_{2,s}T_s + \frac{a_{3,s}T_s}{2} + a_{4,s}T_s^2 + \frac{a_{5,s}}{3}T_s^3 + \frac{a_{6,s}}{4}T_s^4 + a_{7,s}
\]

(2.16)

Below 200 K the specific heat was assumed to be constant. The constants \( a_{1,s} \) to \( a_{7,s} \) were obtained from the NASA CEA software database of Gordon and McBride [57]. The rate of production and destruction of an individual species by chemical reactions is included in the chemical source term \( \omega_s \). A chemical reaction can be expressed as:

\[
\sum_s a_{s}^f X_s \rightarrow \sum_s a_{s}^b X_s
\]

(2.17)

Where \( X_s \) is the species symbol, \( a_{s}^f \) and \( a_{s}^b \) are the stoichiometric coefficients, and \( k_f^s \) and \( k_b^s \) are the forward and backward reaction rates. The chemical source term was computed from the law of mass action:

\[
\omega_s = M_s \sum_s (\beta_{s}^f - \alpha_{s}^f) \left[ k_f^s \prod_s \left( \frac{\rho_s}{M_s} \right)^{a_{s}^f} - k_b^s \prod_s \left( \frac{\rho_s}{M_s} \right)^{a_{s}^b} \right]
\]

(2.18)

The forward reaction rate was calculated applying the Arrhenius law [62]:

\[
k_f^s = A_r T_s^{\beta_s^f} e^{-\frac{E_{ac}}{RT_s}}
\]

(2.19)

The backward reaction rate was obtained from the equilibrium constant:

\[
k_b^s = \frac{k_f^s}{K_{eq}^s}
\]

(2.20)

A detailed overview of the thermo–chemical modeling for reacting flows in chemical non-equilibrium used in the DLR-TAU-code is given by Karl [63]. In the end a dedicated transport equation is solved for each individual chemical species.

For the present investigation, TAU is applied to solve the Reynolds-averaged Navier-Stokes equations on two-dimensional and axisymmetric hybrid structured/unstructured grids. Turbulence modeling is conducted using the Spalart-Allmaras turbulence model [64], which yielded good results for the prediction of separation positions in over-expanded
D. Schneider, K. Kostyrkin, R. Stark, C. Génin & M. Oschwald

Figure 3. Numerical computation domain for the dual-bell nozzle investigation.

The numerical simulations are carried out prior to the tests, without any test data for validation. The numerical setup is based on the authors expertise in dual-bell nozzle transition simulations [39, 44]. The simulations are performed on a two-dimensional and axisymmetric hybrid (structured/unstructured) grid. Figure 3 illustrates the computational domain and the applied boundary conditions. The dimensions of the computational domain are $110 \cdot L_x$ in x-direction by $70 \cdot L_z$ in z-direction, to minimize the influence of the applied flux based far-field boundary conditions [69]. The gas composition in the far-field is assumed to be dry air with a temperature of 300 K. A velocity of 10 m/s in

nozzle flows [65] and the prediction of the operation mode transition of dual-bell nozzles [39, 44]. For discretization of the inviscid flux terms the AUSMDV [66] (Advection Upstream Splitting Method) upwind scheme is applied. A MUSCL-type (Monotonic Upstream-Centered Scheme for Conservation Laws) gradient reconstruction is used to achieve second order spatial accuracy. Viscous terms are computed with a second order central scheme. Time integration is performed by an implicit backward Euler method with LU-SGS scheme. Time-accurate solutions are obtained using a Jameson-type dual time-stepping scheme [67] with second order temporal discretization, applying a physical time-step $10^{-6}$ s. For the modeling of the combustion processes in the reactive nozzle flow a finite rate approach with a 19-step reaction mechanism [68] containing 9 chemical species is applied.

2.1.2. Numerical Setup

The numerical simulations are carried out prior to the tests, without any test data for validation. The numerical setup is based on the authors expertise in dual-bell nozzle transition simulations [39, 44]. The simulations are performed on a two-dimensional and axisymmetric hybrid (structured/unstructured) grid. Figure 3 illustrates the computational domain and the applied boundary conditions. The dimensions of the computational domain are $110 \cdot L_x$ in x-direction by $70 \cdot L_z$ in z-direction, to minimize the influence of the applied flux based far-field boundary conditions [69]. The gas composition in the far-field is assumed to be dry air with a temperature of 300 K. A velocity of 10 m/s in
x-direction is applied in the far-field to stabilize the simulations. The ambient pressure is set to 0.1 MPa.

The nozzle inlet is modeled as a reservoir pressure boundary condition. The total pressure and density as well as the mass fraction of the combustion gas are specified, and the corresponding flow conditions are computed with isentropic flow expansion from the total conditions toward the local flow velocity. The processes in the combustion chamber are neglected and the flow is assumed to enter the convergent part of the nozzle in chemical equilibrium. Based on the total combustion chamber pressure and the mixture ratio, the mass fraction and the total density are computed using the NASA CEA [57] (Chemical Equilibrium with Application) tool. Different values of NPR are reached by adjusting the combustion chamber pressure at constant ambient pressure.

Along the inner nozzle wall a turbulent boundary layer is assumed. Based on former DLR hot-flow nozzle investigations [70] an isothermal temperature profile is applied along the inner nozzle wall. Prismatic sublayers are used to resolve the viscous and thermal boundary layers along the wall. A dimensionless wall spacing $y^+ = O(1)$ is applied to ensure a sufficient resolution of the laminar sublayer and the thermal boundary layer. A density gradient based indicator is used to control the fully automated grid adaptation in the vicinity of shocks and shear layers. The grid resolution is adopted from former conducted detailed grid studies [39,44]. These grid studies have shown that three adaptation steps with a minimum cell edge length of 0.1 mm are sufficient to reach grid convergence.

For the film injection a supersonic inflow boundary condition is applied. The design injection Mach number is $M_{d,f} = 2.57$ at a constant total temperature of $T_{t,f} = 400$ K. Film injection pressure is set either according to predefined film mass flux or dynamically adjusted during the simulations with constant mass flux ratio.

### 2.2. Operation Mode Transition

#### 2.2.1. Baseline Configuration

In a first step, time-accurate URANS simulations between nozzle pressure ratios (NPR) 45 and 95 are performed with the DBN baseline configuration. The baseline configuration features a cooling film mass flow rate of $\dot{m}_f = 185$ g/s and a combustion chamber mixture ratio of $\phi_f = 6.0$. Figure 4(a) illustrates the flow separation position dependence on the nozzle pressure ratio. A clear sea level operation mode can be observed between NPR 45 and 55. For these NPRs the flow separates from the wall in the vicinity of the contour inflection at $x/R_{th} = 7.2$. Beginning at NPR 55 the sneak transition [4] starts and the flow separation position moves into the part of the nozzle extension with negative wall pressure gradient. Between NPR 70 and 85 the actual operation mode transition takes place. If the curve inflection criterion [34] is applied on the flow separation evolution curve, the operation mode transition nozzle pressure ratio is determined to NPR$_{tr} = 80.5$. Between NPR 85 and 95 the dual-bell nozzle operates in clear altitude mode and the flow separates at the end of the nozzle extension.

Figure 5 shows the Mach number distribution of the dual-bell nozzle flow field in sea level (top) and altitude (bottom) operation mode. This corresponds to NPR Values of 45 and 95. The shock pattern shows the typical appearance for dual-bell nozzles with truncated ideal base nozzle contour (see fig. 1). A deformation of the Mach disc can be observed for sea level and altitude operation mode. This flow behavior was observed by former experimental and numerical studies of Génin et al. [71] in 2017. A maximum
Mach number in the flow field of approx. 4.5 is reached for the DBN in altitude operation mode.

Figure 4(b) depicts the velocity of the separation position dependent on the nozzle pressure ratio. The flow separation velocity for a particular NPR is calculated by the following equation:

$$u_{sep}(NPR) = \frac{x(NPR) - x(NPR - 0.5)}{t(NPR) - t(NPR - 0.5)}$$  \hspace{1cm} (2.21)

The flow separation position velocity is almost zero for the DBN in sea level mode. Then it increases until transition NPR is reached. At higher NPR values it decreases again until the steady state of altitude mode is achieved. It can be observed that the maximum velocity of the separation position is reached for the transition NPR obtained by the curve inflection criterion. The maximum velocity occurs at a DBN extension length of approx. $x/R_{th} = 13$, which corresponds to 75% extension length. However, the separation position velocity does not change significantly between 50% and 75% of the extension length. Thus, it can be argued that the maximum separation point velocity
is already reached at 50 % extension length. This is in good agreement with former experimental [41] and numerical [39, 72] studies. The evaluation of the simulation data yields a sneak transition duration of 10.09 s. The total transition duration is determined to be 13.06 s.

Figure 6 illustrates the hysteresis behavior of the investigated dual-bell nozzle. The hysteresis behavior is the difference in flow separation position at the same NPR for the start-up and shut-down of an engine with a dual-bell nozzle. This behavior ensures a stable operation in altitude mode against combustion chamber pressure and ambient pressure fluctuations. The hysteresis shown in figure 6 is obtained by an URANS simulation with negative total combustion chamber pressure gradient beginning at NPR 95. As long as the DBN operates in altitude mode between NPR 95 and 85 there is almost no difference in flow separation position. Beginning at NPR 85 the flow separation position evolution curves diverge. Applying the curve inflection criterion, the re-transition nozzle pressure ratio is determined to NPR$_{ret}$ = 60.5. With the definition of the hysteresis [39], the margin between transition and retransition NPR is 25.5 %. The maximum flow separation position velocity during retransition is approx. 40 m/s. It is reached at approx. $x/R_{th} = 10$, which is a little bit upstream of 50 % extension length. Thus, compared to the up-ramping simulations the maximum flow separation velocity of the down-ramping simulation is twice as high.

2.2.2. Impact of Mixture Ratio Variation

The impact of the combustion chamber mixture ratio on the dual-bell operation mode transition behavior is studied in this section. Therefore, URANS simulations of the DBN start-up between combustion chamber mixture ratios of $o/f = 5.0$ and $o/f = 7.0$ are carried out. Figure 7(a) illustrates the flow separation position evolution curves for the baseline configuration and the configurations with $o/f = 5.0$ and $o/f = 7.0$. A clear impact of the combustion chamber mixture ratio on the separation position evolution curve can be observed. Increasing the mixture ratio leads to a shift of the operation mode transition nozzle pressure ratio to lower values. This is in good agreement with former experimental [41] and numerical [44] studies of authors using LOX/CH$_4$ as propellant combination. The transition NPR is 77.5 for a combustion chamber mixture ratio of 7.0.
346 D. Schneider, K. Kostyrkin, R. Stark, C. Génin & M. Oschwald

(a) Impact of the combustion chamber mixture ratio on the dual-bell transition behavior. (b) Combustion chamber mixture ratio impact on the flow separation position velocity.

Figure 7. Combustion chamber mixture ratio impact on the dual-bell operation mode transition behavior.

Decreasing the combustion chamber mixture ratio to 5.0 leads to a shift of the transition NPR to 83.5.

Figure 7(b) shows the flow separation position velocity as a function of o/f for the three investigated mixture ratios. It can be observed that with decreasing o/f the maximum flow separation position velocity increases. For an o/f of 7.0 a maximum flow separation position velocity of approx. 19 m/s is reached. Decreasing the o/f to 5.0 the maximum flow separation position velocity increases to approx. 26 m/s.

2.2.3. Impact of Cooling Film Mass Flux Variation

In this section the influence of the cooling film mass flux on the DBN transition behavior is studied. Therefore, URANS simulations at constant o/f = 6.0 of the DBN start-up process are performed varying the cooling film mass flow rate between 135 g/s and 235 g/s. Figure 8(a) illustrates the comparison of the flow separation evolution curves of the three investigated configurations. It can be observed, that an increase of the cooling fluid mass flow rate leads to a shift of the transition nozzle pressure ratio to lower val-

(a) Impact of the cooling film mass flow rate on the dual-bell transition behavior. (b) Cooling film mass flow rate impact on the flow separation position velocity.

Figure 8. Cooling film mass flow rate impact on the dual-bell operation mode transition behavior.
ues. For the case of a film mass flow rate of \( m_f = 235 \text{ g/s} \) a transition NPR of 78.5 is determined. A decrease of the cooling film mass flow rate to \( m_f = 135 \text{ g/s} \) leads to an increase of the transition NPR to 81.5.

The flow separation position velocity dependence on the nozzle pressure ratio is illustrated in figure 8(b) for the three different cooling film mass flow rates. It can be observed that an increase of the cooling film mass flow rate yields a decrease of the maximum flow separation position velocity. A maximum flow separation position velocity of approx. 19 m/s is reached for a cooling film mass flow rate of \( m_f = 235 \text{ g/s} \). A decrease of the cooling film mass flow rate to \( m_f = 135 \text{ g/s} \) leads to an increase of the maximum flow separation position velocity to approx. 24 m/s.

2.2.4. Impact of Mass Flux Ratio Variation

In this section a simultaneous o/f and cooling film mass flux variation is conducted. Therefore, the total hydrogen mass flux \( m_{H_2,t} \) is split into a combustion chamber part \( m_{H_2,c} \) and a cooling film part \( m_f \). It is assumed that the total hydrogen \( m_{H_2,t} \) and the oxygen \( m_{O_2} \) mass fluxes are constant for a rocket engine operating in steady state conditions. This means that an increase of the cooling film mass flow rate \( m_f \) would lead to a simultaneous increase of the combustion chamber mixture ratio o/f. It was shown before that both effects result in a shift of the transition nozzle pressure ratio to lower values. Thus, the dual-bell operation mode transition could be triggered by redirecting mass flow from the combustion chamber to the cooling film.

For the following investigations the mass flux ratio \( \lambda \) is introduced and kept constant during the simulations:

\[
\lambda = \frac{m_f}{m_{H_2,c} + m_{O_2}} \quad (2.22)
\]

The reference point for the performed simulations is chosen to be at \( \text{o/f} = 6.0 \) and \( \lambda = 3.0 \% \). Readjusting the combustion chamber mixture ratio to \( \text{o/f} = 7.0 \) yields a mass flux ratio of \( \lambda = 5.2 \% \).

Figure 9(a) illustrates the comparison of the DBN operation mode transition between the baseline configuration, the configuration \( \lambda = 3.0 \% \) and the configuration \( \lambda = 5.2 \% \). It can be observed that the coupling of the cooling film mass flux and the main mass flux to a constant ratio of \( \lambda = 3.0 \% \) has a slight higher transition NPR compared to the baseline configuration with constant cooling film mass flow rate. The operation mode transition NPR is determined to \( \text{NPR}_{tr} = 81.5 \) for the \( \lambda = 3.0 \% \) configuration. An increase of the mass flux ratio to \( \lambda = 5.2 \% \) leads to a shift of the transition NPR to \( \text{NPR}_{tr} = 77.4 \). This corresponds to a reduction of 5 % relative to the \( \lambda = 3.0 \% \) configuration. Thus, a simultaneous adjustment of the combustion chamber mixture ratio and the cooling film mass flow rate is an effective way to actively control the dual-bell operation mode transition.

Figure 9(b) depicts the flow separation position velocity as a function of NPR for the three investigated configurations. The coupling of the cooling film mass flux and the main mass flux to a constant ratio of \( \lambda = 3.0 \% \) leads to a slight higher maximum flow separation position velocity of approx. 26 m/s compared to the baseline configuration. An increase of the mass flux ratio to \( \lambda = 5.2 \% \) yields a maximum flow separation position velocity of approx. 23 m/s.
2.2.5. Cooling Film Triggered Dual-Bell Operation Mode Transition

Another way to trigger the dual-bell operation mode transition is to increase the cooling film mass flow rate at constant combustion chamber conditions. An URANS simulation at NPR 55 with cooling film ramping is conducted to evaluate this concept. The film ramping is initiated at a cooling film nozzle pressure ratio (ratio between total cooling film pressure and ambient pressure) of NPR\(_{f} = 35\). The cooling film nozzle pressure ratio is increased until the transition to altitude operation mode occurs.

Figure 10(a) illustrates the flow separation evolution curve as a function of the cooling film nozzle pressure ratio. It can be observed that the flow separation position remains in the vicinity of the contour inflection until a cooling film pressure ratio of NPR\(_{f} = 75\) is reached. A further increase of the cooling film pressure ratio leads to the operation mode transition with a very steep flow separation evolution curve. The application of the curve inflection criterion yields a transition cooling film pressure ratio of NPR\(_{f, tr} = 80.5\). After transition to altitude operation mode is completed, the flow separation position remains approx. 6 % upstream compared to the former investigated configurations.

Figure 10(b) depicts the flow separation position velocity as a function of the cool-
cooling film pressure ratio for the investigated configuration. It can be observed that the maximum flow separation position velocity is significantly higher compared to all former investigated configurations. It is reached at approx. 50% of the extension length and has an absolute value of $83.34 \text{ m/s}$. Compared to the maximum flow separation position velocity of the baseline configuration, the flow separation position is 3.85 times faster during operation mode transition. Thus, a significantly shorter transition duration compared to the baseline configuration is achieved.

Cooling Film ramping is an effective method to actively trigger and control the dual-bell operation mode transition independent of the main combustion chamber conditions. The results are obtained for a relatively low main NPR of 55. A higher starting main NPR would result in a lower cooling film pressure ratio $\text{NPR}_{f}$ needed to trigger the operation mode transition.

3. Conclusions

A numerical study was conducted to investigate the impact of a film-cooled dual-bell nozzle extension on its operation mode transition behavior. Therefore, unsteady Reynolds-averaged Navier-Stokes simulations of the transition process between sea level and altitude mode were carried out. In a first step the operation mode transition and retransition of a baseline configuration was investigated. A transition nozzle pressure ratio of 80.5 and a hysteresis margin of 25.5% were predicted for the following hot-flow experiments at DLR Lampoldshausen.

Subsequently, the impact of the combustion chamber mixture ratio on the dual-bell operation mode transition was investigated. It was found that an increase of the combustion chamber mixture ratio leads to a shift of the transition nozzle pressure ratio to lower values. Furthermore, the flow separation position velocity during operation mode transition decreases with increasing combustion chamber mixture ratio.

Afterwards, the impact of the cooling film mass flow rate on the dual-bell operation mode transition was studied. The investigation yielded a shift of the transition nozzle pressure ratio to lower values with increasing cooling film mass flow rate. However, the flow separation position velocity decreases with increasing cooling film mass flow rate.

A combination of the two former investigated effects was studied, to introduce an active control method for the dual-bell operation mode transition. Therefore, simulations with constant mass flux ratios between cooling film mass flux and total combustion chamber mass flux were performed. Then, a variation of this mass flux ratio was conducted. Increasing the mass flux ratio by redirecting hydrogen mass flow from the combustion chamber to the cooling film leads to an increase of the combustion chamber mixture ratio. The higher cooling film mass flow rate and the higher combustion chamber mixture ratio shift the transition nozzle pressure ratio to lower values. Thus, the simultaneous adjustment of the combustion chamber mixture ratio and the cooling film mass flow rate was found to be an effective way to actively control the dual-bell operation mode transition.

Cooling film mass flow rate ramping was found to be another method to actively control the dual-bell operation mode transition, independent of the combustion chamber conditions. It was shown that triggering the operation mode transition to altitude mode is feasible by only increasing the cooling film mass flow rate. The flow separation position velocity during operation mode transition was found to be significantly higher compared to the baseline configuration.
Acknowledgments

The present work was conducted in the framework of the German Aerospace Center (DLR) project TAUROS (TAU for Rocket Thrust Chamber Simulation) focusing on the qualification and advancement of the DLR flow solver TAU for liquid rocket thrust chamber applications. The financial support of the DLR Space Research Programmatic is highly appreciated. Financial support has been provided by the German Research Foundation (Deutsche Forschungsgemeinschaft) in the framework of the Sonderforschungsbereich Transregio 40.

References


Operation Mode Transition of Film-Cooled Dual-Bell Nozzles


