DOI: 10.13009/EUCASS2019-515

DLR Project TAUROS: TAU for Rocket Thrust Chamber Simulation

Dirk Schneider†, Justin Hardi, Jan Martin, Ralf Stark, Dmitry Suslov, Federica Tonti,
Jan van Schyndel, Victor Zhukov and Michael Oschwald
German Aerospace Center (DLR)
Institute of Space Propulsion, Rocket Propulsion Department
Im langen Grund, 74239 Hardthausen, Germany
†dirk.schneider@dlr.de

Eugeni Costa Ruiz
German Aerospace Center (DLR)
Institute of Space Propulsion, Engineering Department
Lampoldshausen, 74239 Hardthausen, Germany

Stefan Fechter, Tim Horchler, Sebastian Karl and Klaus Hannemann
German Aerospace Center (DLR)
Institute of Aerodynamics and Flow Technology, Spacecraft Department
Bunsenstr. 10, 37073 Göttingen, Germany

Marko Alder, Sebastian Jack and Thino Eggers
German Aerospace Center (DLR)
Institute of Aerodynamics and Flow Technology, Spacecraft Department
Lilienthalplatz 7, 38108 Braunschweig, Germany

Ansgar Lechtenberg and Peter Gerlinger
German Aerospace Center (DLR)
Institute of Combustion Technology, Numerical Simulation Department
Pfaffenwaldring 38-40, 70569 Stuttgart, Germany

Abstract

The present work gives a brief overview of the DLR (German Aerospace Center) project TAUROS (TAU for Rocket Thrust Chamber Simulation). The DLR project TAUROS is a collaboration between three different DLR institutes, and is funded by the DLR Space Research Programmatic. The project focuses on the qualification and advancement of the DLR flow solver TAU for liquid rocket thrust chamber application. The main objective of TAUROS is to ensure an independent DLR competence in the numerical modeling of space propulsion systems and their components. The collaboration bundles the DLR competences in the field of development and application of numerical methods for liquid space propulsion systems, and in the field of development and operation of research rocket thrust chambers. This ensures a verification and validation of the developed numerical methods independent from external sources. In detail the project addresses the strong coupling between reactive flows, thermal environment, and structure loads. Therefore, the project is split into the main work packages combustion chamber, thrust nozzle, thrust chamber wall cooling, and test facilities. The highlights of the achieved progress in these work packages are described in the present publication.

1. Introduction

The development of rocket engines and of the associated test facilities has to be complemented by appropriate tools. Beside of analytical and empirical methods numerical simulations can significantly reduce the duration and costs of the design, the development, the test and the evaluation process. Because of the increase of available computing

DOI: 10.13009/EUCASS2019-515

Figure 1: Logo of the DLR project TAUROS

capacity, the numerical simulation will evolve into a substantial design tool for future space transportation engines. The reliable prediction of the reactive flow behavior, the thermal environment and the structure loads of future rocket engine components and the entire system requires a permanent enhancement and adjustment of the numerical methods. Therefore, the DLR project TAUROS aims on the qualification and advancement of the space transportation extension of the DLR flow solver TAU.

TAUROS is based on the former DLR project ProTAU[38] (Rocket Propulsion in TAU), which was successfully completed in 2016. Collaborators of the project are the Institute of Space Propulsion (RA), the Institute of Aerodynamics and Flow Technology (AS) and the Institute of Combustion Technology (VT). Within the project the expertise in the development and application of the DLR-TAU code[21] in the field of liquid rocket engines (AS), in the development of high order numerical methods for combustion processes (VT), and in the operation and development of test facilities, research rocket engines and their components (RA) is combined in a close collaboration. The project has started in 2017 and will be finished at the end of 2019. The project is divided into four main work packages:

- Combustion chamber
- Thrust nozzle
- Test facilities
- Thrust chamber wall cooling

The present work gives a brief overview of the conducted work within these main work packages. During the project the research results gained within the individual work packages are published at scientific conferences and in research journals. For a more detailed insight into the project, the reader is referred to these publications.

2. Combustion Chamber

The physical mechanisms inside rocket combustion chambers involve many complex aspects like multiphase flows, spray effects, chemical reactions, turbulence-flame interaction and acoustic effects leading to instabilities, to name a few. Each of the aspects poses great challenges to numerical modeling and deserves a project on its own. One goal of TAUROS is to have a good compromise between covering all of the aforementioned aspects reliably and accurately while keeping the modeling effort at a reasonable cost. To assess the accuracy of the applied models, simulations are accompanied by experimental validation and high-order numerical simulations. As a consequence, this project uses the well documented results from the experimental campaigns of DLR Lampoldshausen. This work is divided into several activities which are briefly described in the following subsections.

DOI: 10.13009/EUCASS2019-515

2.1 Extension of TAU Solver for Reacting Cryogenic Flows

The focus of this activity is the extension of the DLR-TAU solver to handle cryogenic combustion processes in rocket thrust chambers. The baseline TAU solver for reactive gas flows is extended to use accurate high-pressure equations of state (EOS) as well as a flamelet combustion model for complex kinetic reaction schemes. Our aim is to extend TAU to provide a validated RANS based solver allowing for a detailed analysis of combustion chamber flows at low to moderate computational cost. This tool is able to accurately predict the most crucial flow properties needed for combustion chamber design, such as wall heat fluxes, chamber pressure as well as flow structure and heat loads in the nozzle.

Rocket combustion chambers are typically operated at cryogenic conditions with the oxygen being injected at low temperatures. At these conditions, the ideal gas EOS cannot predict the fluid behavior of liquid oxygen at the injection conditions correctly. Thus, the TAU solver is extended by an advanced fluid model able to cope with cryogenic states up to high temperatures within the flame. Therefore, several mixture models are assessed ranging from the Multi-Fluid-Mixing model developed by Banuti et al.[1] to cubic EOS mixture rules (for a description see e. g. Poling et al.[18]). The aim is to understand the influence of the thermodynamic mixture model on the global combustion chamber design points, e. g. wall heat fluxes, combustion chamber pressure as well as flame lengths. To efficiently use complex EOS within the flow solver, a quadtree-based tabulation algorithm is used (Fechter et al.[6]). The influence of the EOS on the simulation of the BKC combustion chamber is summarized in Fechter et al.[7] . These extensions allow to reproduce the wall heat flux of a realistic combustion chamber like the 49 injector BKD test case (see Figure 2) successfully. Our tool also resolves the complex flame interaction between the injectors that have been arranged on three concentric rings.

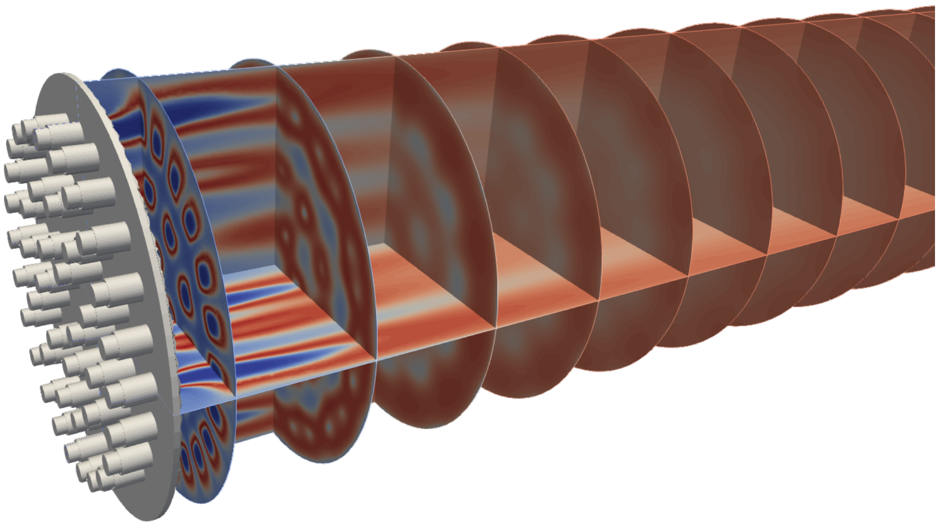


Figure 2: BKD combustion chamber simulation at a nominal combustion chamber pressure of 4.9 MPa

The continuing development of rocket combustors is currently targeted at replacing H_2 as fuel due to its difficult manageability. One promising alternative is methane because of its good storability and ease of handling. From a modeling point of view, using CH_4/O_2 unfortunately increases the computational cost drastically as the application of much larger reaction mechanisms like the GRI-3.0 is required. Therefore, we follow a flamelet approach that can greatly reduce the required computational time by precomputing and storing the flame structure in lookup tables. A stand-alone flamelet solver named RGFlamelet for the mixture fraction based flamelet equations is implemented and validated. It allows to efficiently compute large sets of counter flow diffusion flames with various thermodynamic models (ideal gas, Single-Fluid- and Multi-Fluid-Mixing models) and different reaction mechanisms including pressure-dependent reactions. A further increase in the computational speed can be achieved by storing cubic mixing coefficients and gas transport properties in the flamelet table as this avoids the costly evaluation in the solver. Turbulence-chemistry interaction can also be taken into account efficiently by an assumed β-PDF-model.

RGFlamelet has been validated against the publicly available flamelet solver FlameMaster[17] and literature results for flames under supercritical injection conditions.[14,15] The flamelet approach has been successfully applied to the 7-injector CH_4-O_2 combustor experiment from TU Munich.[25] The simulated chamber pressure and wall heat fluxes are in good agreement with the results from the other test case participants and the experimental result.[16] Compared to a detailed chemistry approach, the flamelet model speeds up this calculation by a factor of 550.

DOI: 10.13009/EUCASS2019-515

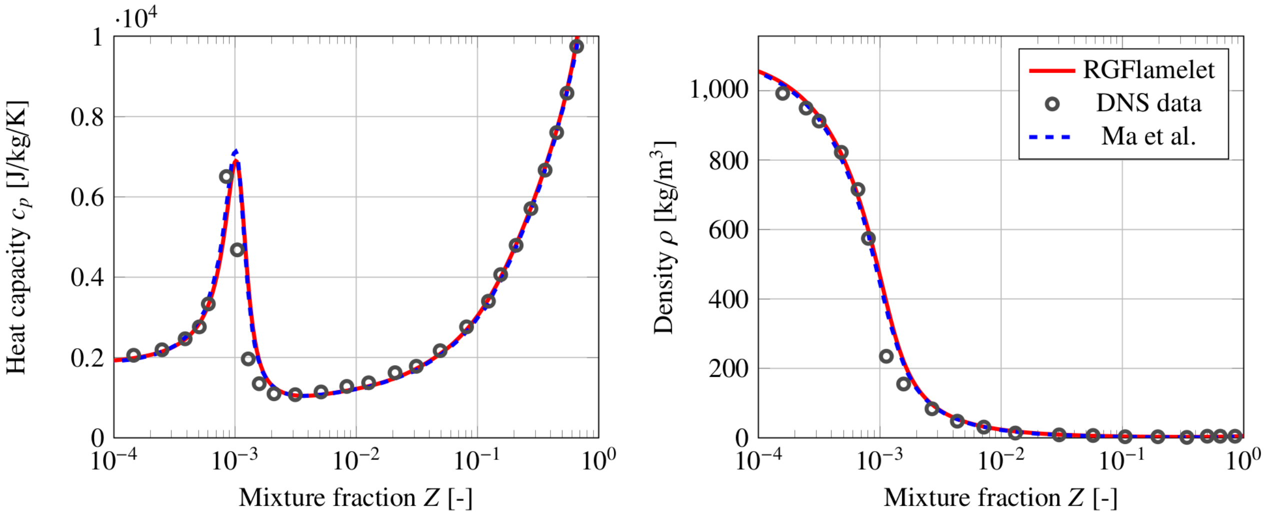


Figure 3: Validation of the real gas flamelet solver RGFlamelet using results from Ma et al.[15] and DNS data from Lacaze et al.[14] The plot shows the flame structure for a H_2/O_2 counter flow diffusion flame with transcritical oxygen injection at 7 MPa.

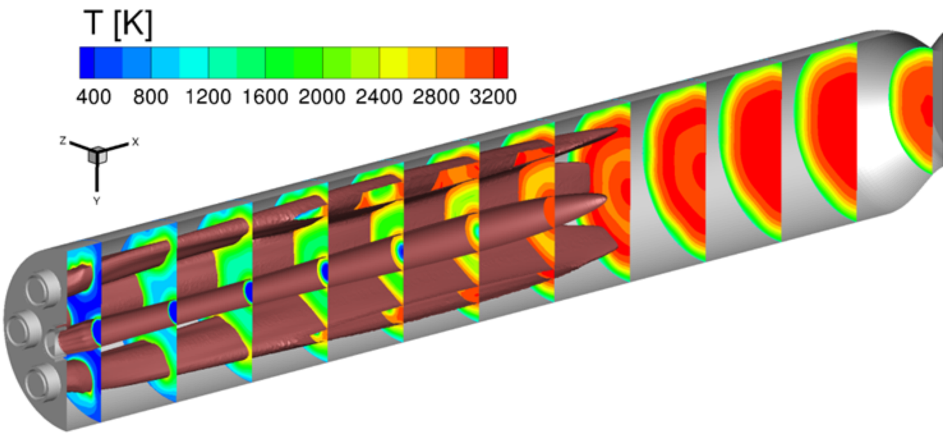


Figure 4: TAU results for the Munich CH_4-O_2 combustor test case from the SFB-TRR-40 summer program 2017.

2.2 Design of a Single Injector Combustion Chamber for Numerical Validation Purposes

This work package aims at designing a single-injector experiment for validation purposes of TAU simulations with LOx/Methane combustion. Due to the demand of gaining experimental data which could be used for validation purposes several requirements can be derived, but not all of them are compatible with each other. On one hand optimal boundary conditions and ideally measurements in any location along the chamber wall and even inside the flame are desired, but on the other hand representative geometry and operating conditions should be kept. In this way the relevance of the generated validation data, and therefore the proof of applicability of the validated code in simulating combustion chamber dynamics of interest should be maintained.

A LOx/Methane combustion system which can operate above the LOx critical pressure is required. Furthermore, a modular design for varying the injection conditions is considered to be essential. The capability of validating the thermodynamic values calculated by the code shall be given by a high sensor density along the chamber axis. Validating the flame's shape and length could be supported by an optical access which length ideally matches the length of the intact LOx core.

The preliminary design is illustrated in Fig. 5. The figure shows a possible design which includes the demands presented in the previous subsection. Temperature, pressure and unsteady pressure sensors are alternately distributed along the chamber axis. The optical access and its preliminary dimensions are modeled with transparent parts. Finally the injector head with its measurement devices can be seen at the top left part of the image and the nozzle at the bottom right. The design is currently being subjected to analysis of thermal and mechanical loads.

DOI: 10.13009/EUCASS2019-515

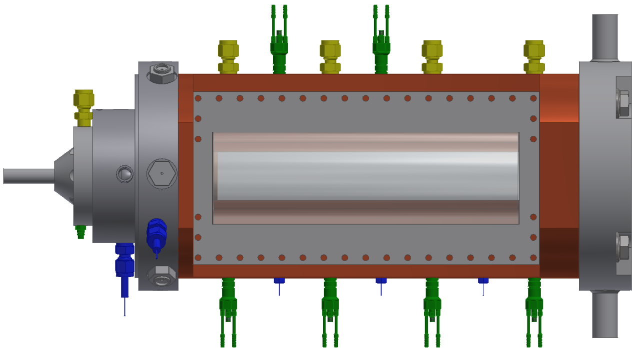


Figure 5: Preliminary design of a combustion chamber for numerical validation purposes

2.3 Reduced Reaction Kinetics of CH_4/O_2 Mixtures

Flamelet combustion models have a limited validity region. They require the assumption of an infinitely fast chemistry. Some flamelet models also assume the turbulence-chemistry interaction. However, this interaction means the interaction of mixing with turbulence, while the chemical timescale is infinitely small. Earlier, the validity of the flamelet approach was discussed in work.[43] In general, the assumption of an infinitely fast chemistry holds in rocket engines due to high temperatures and pressures. However, it is not valid in the shear layer of cryogenic propellants and near walls. The expansion of gas in rocket nozzle shifts the chemical equilibrium; therefore, it is necessary to take into account reaction kinetics in modelling rocket nozzle flows.[40,43] The finite rate chemistry assumption does not have

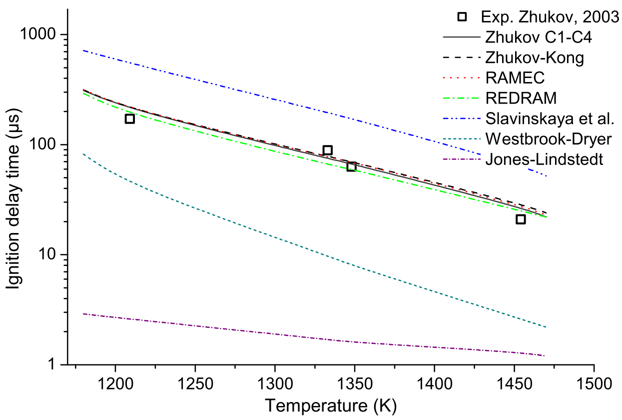


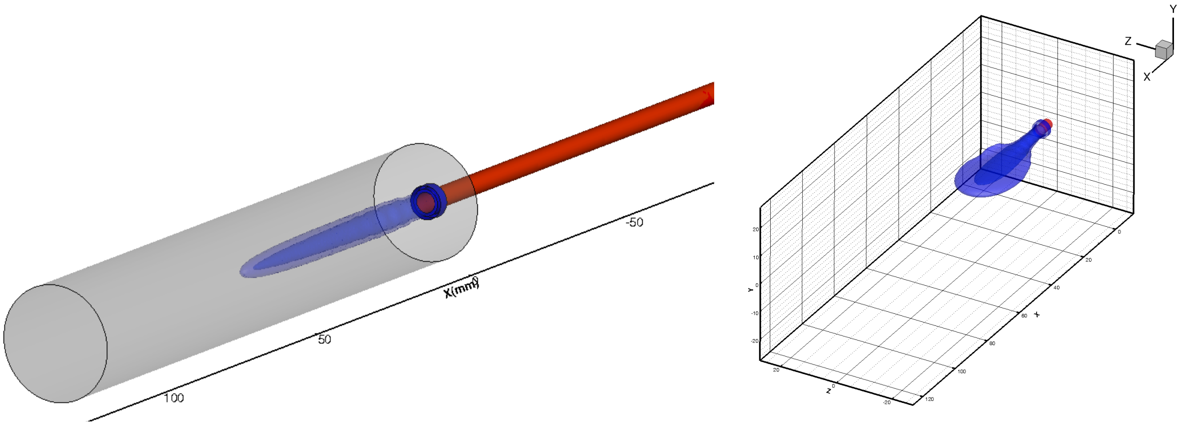
Figure 6: Comparison of different methane mechanisms with experimental data:[42] methane–air, ϕ = 0.5, 150 atm.

the validity limitations of the flamelet models but requires modeling of reaction kinetics using reaction mechanism. Hydrogen rocket engines are generally simulated using the finite rate chemistry (FRC) model with a detailed reaction mechanism. However, methane detailed reaction mechanisms have approximately ten times more reactions and species than hydrogen mechanisms. This makes it impossible to simulate methane combustion chambers using detailed reaction mechanisms. In the frameworks of the TAUROS project, this problem is solved by two means. The first way is the implementation of flamelet model into the TAU code, as it was already described in the previous section. The second way is the development of a reduced methane kinetic mechanism.[41] The developed mechanism is a skeletal mechanism and consists of 23 species and 51 reactions. The new mechanism retains the accuracy of the parent mechanism Zhukov C1-C4[39] in ignition delay times (see Fig. 6) and in temperatures of CH_4/O_2 flames. However, it is not very accurate in laminar flame speed and flame temperatures at high strain rates, and this is a matter of future DLR projects.

2.4 Combustion Instabilities

In the frame of this work package, the study of combustion instabilities in combustion chambers is conducted through numerical simulations of flame-acoustics interaction. This work package, in particular, focuses on the simulation of the experimental combustor BKH operated at the DLR Institute of Space Propulsion in Lampoldshausen[11] using a

DOI: 10.13009/EUCASS2019-515



(a) Longitudinal mode excitation (b) Transverse mode excitation

Figure 7: Isosurfaces at ρ= 10 kg/m^3 and 100 kg/m^3 for different acoustic excitation conditions

URANS approach in TAU. Single injector configurations for different excitation conditions are studied, representing a flame either in a longitudinal or transverse acoustic field, in order to subsequently perform a full chamber simulation, including more realistic excitation[35,36] and boundary conditions.[37]

The main goal of the simulation of a single injection element of BKH is having a first understanding of the dynamics of the flame, in order to build a Flame Describing Function (FDF) involving different operating points, and use then these results for the comparison to the full chamber simulation and to the experiment, in order to know to what extent it is possible to extrapolate information from a simplified configuration to correlate to a more complex one.

Two excitation conditions with different configurations have been simulated: pressure excitation (longitudinal mode) and transverse velocity excitation (transverse mode). For the pressure excitation case, the simulations reproduce the formation of wave-like structures on the shear layer at the interface between the streams of LOx and H_2, as shown in Figure 7 (a). For transverse velocity excitation case, the shortening and flapping of the LOx core is qualitatively well captured, as shown in figure 7 (b).

2.5 High Order Methods

The numerical simulation of combustion instabilities is still challenging task. Especially the investigation of weak oscillating combustion chambers is difficult if there is too much numerical dissipation which may be caused by low order discretization or URANS (Unsteady Reynolds Averaged Navier-Stokes) turbulence models. In this part of the TAUROS project, a weak oscillating operation point of the experimental single injector combustion chamber BKC was simulated. The BKC was designed to study combustion oscillations of cryogenically injected propellants at high pressure conditions. The experiment of this particular test case was performed by Smith *et al.*[26] Numerical studies by Seidl[23] with 2D-URANS showed that self-exited combustion chamber instabilities could not be predicted. Therefore the same operation point is simulated again with high order IDDES (Improved Delayed Detached Eddy Simulation).

In a IDDES the near wall region is treated like in RANS simulation while the core flow is simulated by LES (Large Eddy Simulation). Switching takes place automatically using empirical functions depending on wall distance, grid size and the flow field.[24] The spatial discretization is up to 6^{th} order using the MPL^{ld} (multi-dimensional limiting

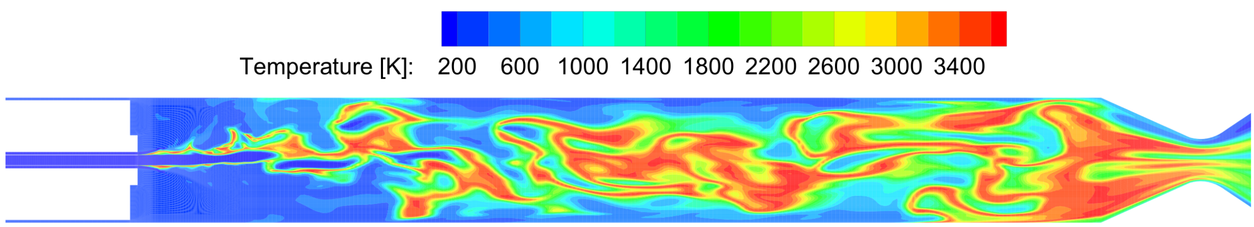


Figure 8: Simulated instantaneous temperature distribution of the combustion chamber BKC calculated with high order methods.

DOI: 10.13009/EUCASS2019-515

process, low diffusion[10]) scheme. MLP^{ld} uses information from diagonal volumes of a discretization stencil for an improved shock resolution and improved convergence compared to conventional TVD (total variation diminishing) limiters.[10] The temporal discretization is realized by a 2nd order BDF (backward differentiation formula) method combined with a LU-SGS (Lower-Upper symmetric Gauss-Seidel) scheme.[30] The time-accurate integration is done by an implicit dual time stepping method. The combustion processes are modeled in detail by FRC (Finite Rate Chemistry). The turbulence/chemistry interaction is realized by a multi-variate assumed PDF (probability density function) method.[2,9]

The simulated combustion chamber shows good agreement with the experimental studies both in mean and in dynamic pressure. The deviation in mean pressure is about 1.5 %, the frequencies of the longitudinal modes were predicted accurate by the simulation. Fig. 8 shows an exemplary instantaneous temperature distribution of the simulated BKC calculated with the explained high order method.

3. Thrust Nozzle

During the transient startup and shutdown process of a rocket engine's nozzle asymmetrical effects tend to deform the nozzle, and cause side loads that can lead to fatal damage of the engine. Based on previous investigations with truncated ideal contour (TIC) nozzles, the main focus in TAUROS is to simulate the transition of different flow patterns within nozzles using a thrust optimized parabola (TOP) contour. This transition phase between shock- and separation-patterns is known to cause the highest peaks in side loads for the widely used TOP nozzles. Numerical and experimental data are therefore collected and compared to yield a validated method for numerical simulation of these effects. Furthermore, a modified Reynolds stress turbulence model is applied to investigate the dual-bell nozzle operation mode transition behavior under LOx/CH4 hot-flow conditions.

3.1 Experimental Investigation of TOP Nozzles

To study the transition from free shock separation (FSS) to restricted shock separation (RSS) and vice versa, a nozzle is needed featuring reliable and reproducible transitions. For this reason the Volvo S1 sub-scale TOP nozzle was redesigned. The DLR-S1 TOP nozzle was tested at the horizontal test position of the cold flow test facility P6.2 (Fig. 9, left). It was supplied with dry gaseous nitrogen for total pressures up to p_0 = 5.8 MPa. The test specimen was equipped with 3 rows of pressure ports to measure the wall pressures, and the exhaust jet was visualized using a b/w Schlieren setup in z-configuration (Fig. 9, right). The results serve as validation cases for the conducted numerical simulations.

Out of five numerically deformed nozzle geometries, three test specimens were chosen and manufactured. Its undeformed geometries are equally to the DLR-TOP and DLR-S1 geometries, respectively. The DLR-S1 contour was deformed with an oval and a triangular shape (Fig. 10). The results will serve the validation of numerical studies on the FSS to RSS transition behavior in ovalized TOP nozzles.

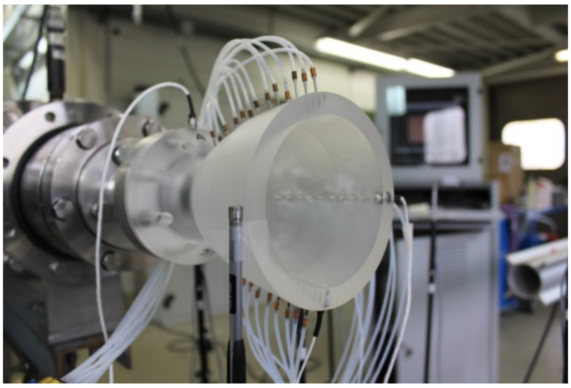


Figure 9: DLR-S1 at horizontal P6.2 test position (left) and related schlieren image of the exhaust jet (right)

DOI: 10.13009/EUCASS2019-515

Figure 10: Ovalized DLR-TOP (left), DLR-S1 triangular shape (middle) and ovalized DLR-S1 (right)

3.2 Simulation of Deformed TOP Nozzles

To evaluate the effect of a given deformation onto the transition behaviour of TOP nozzles, the startup and shutdown process was simulated in five different nozzle geometries, two undeformed base nozzles and three nozzles deformed in oval and triangular shapes (see section 3.1). During the transition phase of the startup of the undeformed nozzles small local zones of flow reattachment formed at the nozzle end, and quickly moved around in circumferential position. These highly instationary and asymmetrical effects cause large side loads that vary in value and direction and match the experimental findings. For the deformed nozzles, the local reattachment in the early transition phase happens at the local minima of the nozzle wall's radius. The changed wall deflection angle and pressure distribution in this area of the wall act as seeding points and attractors. Transition to restricted shock separation patterns is hence triggered earlier than in the undeformed case and, due to the regular but non-rotational-symmetrical shape of these effects, side loads are reduced and stabilized in their direction. Figure 11 shows the axial wall friction coefficient on the wall of the ovalized S1 nozzle. Local flow separation and reattachment lines are indicated by zero crossing (change from red to blue and vice-versa). Before complete reattachment of the flow, the free jet is highly tilted, as seen in Figure 12.

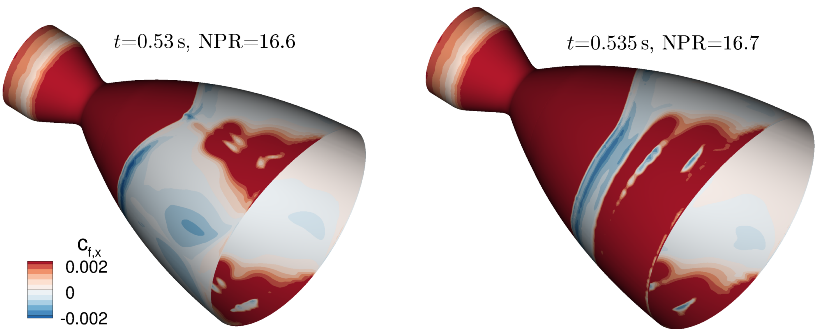


Figure 11: Axial skin friction coefficient on inner wall of ovalized S1 nozzle

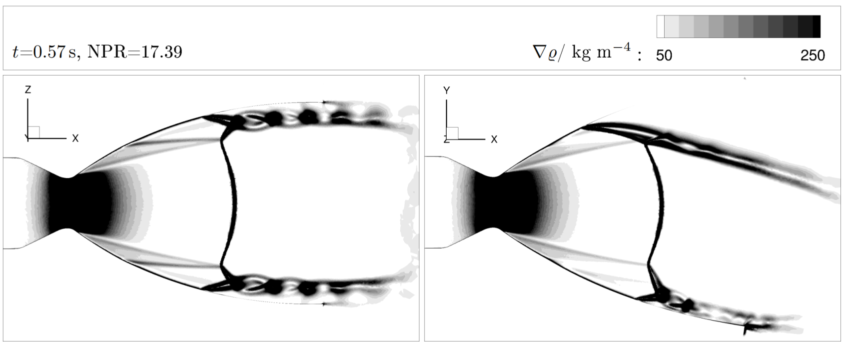


Figure 12: Density gradient (comparable to Schlieren images) in two axial sections through the ovalized S1 nozzle

DOI: 10.13009/EUCASS2019-515

The knowledge gained from the aforementioned simulations as well as the insights from instationary coupled simulations of TIC nozzles are being used to simulate the startup process of a flexible TOP nozzle including fluid structure interaction in the ongoing last project phase.

3.3 Dual-Bell Nozzle Operation Mode Transition

The supersonic flow in rocket nozzles is characterized by high Reynolds numbers. Therefore, resolving the large and middle turbulence scales is often too expensive in terms of simulation duration. Thus, Reynolds-averaged Navier-Stokes (RANS) equations are still solved for most of the technical problems. A general disadvantage of RANS turbulence models is the prediction of the interaction between shocks and turbulence in the flow field. The work package deals with the development of an empirical correction model for Reynolds stress turbulence models.[12] The impact of the correction model is verified by simulations of the dual-bell nozzle flow behavior under LOx/CH4 hot-flow conditions.[20] The dual-bell nozzle is an altitude compensating nozzle concept. It combines the advantage of a nozzle

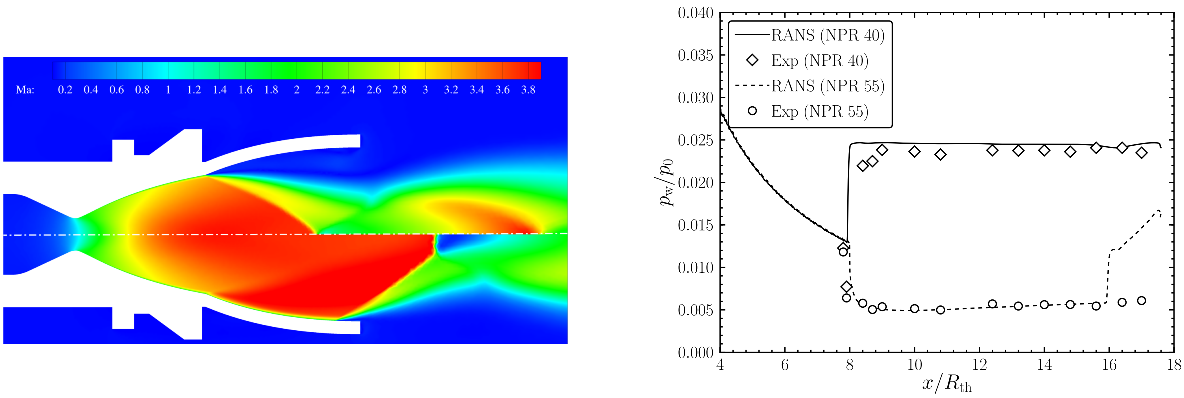


Figure 13: Mach number distribution (left) and related wall pressure profiles (right) of the dual-bell nozzle in sea level and altitude mode

with small expansion ratio under sea level conditions and a nozzle with large area ratio 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 given nozzle pressure ratio (total combustion chamber pressure over ambient pressure), the flow separation position moves abruptly to the exit area of the dual-bell nozzle extension. Thus, a higher expansion ratio is reached for a wide range of the launcher trajectory. This increase of engine performance yields a potential payload gain.[28] Figure 13 illustrates the Mach number distribution of the simulated dual-bell nozzle flow in sea level (top) and in altitude (bottom) mode, and the corresponding wall pressure distributions. It can be observed that the prediction of the dual-bell nozzle flow field, and the wall pressure distribution is significantly improved by the implemented Reynolds stress turbulence model correction compared to former studies of the authors.[19,20]

4. Test Facilities

Acoustic emissions are a key issue during rocket engine starts. The payload of the rocket and the rocket itself are sensitive to high acoustic loads. Therefore, it is very valuable to be able to predict acoustic emission before a rocket engine is started the first time. The acoustic loads on the payload and the rocket depend on some factors like intensity of the acoustic source, the geometry surrounding the source, or the weather. Rocket engine test benches are more robust and less sensitive to high acoustic loads. However, if the test bench is near a populated area, legal acoustic emission restrictions have to be taken into account. In this case the prediction of the acoustic emission of the performed tests is an appreciated information, especially during the licensing process of a new test facility.

The P4.1 is a vacuum simulation test bench for upper stage engines located at the German Aerospace Center in Lampoldshausen. Vacuum test benches have a strong coupling interaction between the supersonic jet of the engine and the supersonic diffusor. This interaction is specially relevant during gimballing tests in vacuum. If a certain degree of inclination is surpassed, the vacuum breaks down abruptly, putting the integrity of the nozzle extension in danger.

DOI: 10.13009/EUCASS2019-515

Being able to predict and to simulate the gimballing envelope during vacuum operation brings many advantages for the operator of the bench and for the test customer.

4.1 Experimental Test Bench Acoustics Investigation

The noise emission of the rocket engine exhaust jet is crucial for the test facilities and their related public environments. Thus, to predict the broadband noise emission as well as prominent jet-generated tones, the exhaust jet's acoustical characteristics must first be understood. A lot of work has been performed on broadband noise emissions of jet engine exhaust jets,[13,31] jet-related specific tones,[33,34] and the interaction with test facilities.[22] Unfortunately, the literature results are limited to jet Mach numbers M_j up to 2.0 and temperature ratios T_0/T_a up to 4, whereas rocket engines feature exhaust jet Mach numbers above 4 and temperature ratios above 11. It is questionable whether the reported jet engine findings can be directly assigned to rocket engines. For this reason, the DLR conducted initial tests to extend the existing literature data base and to generate validation data for numerical simulations.[3,4]

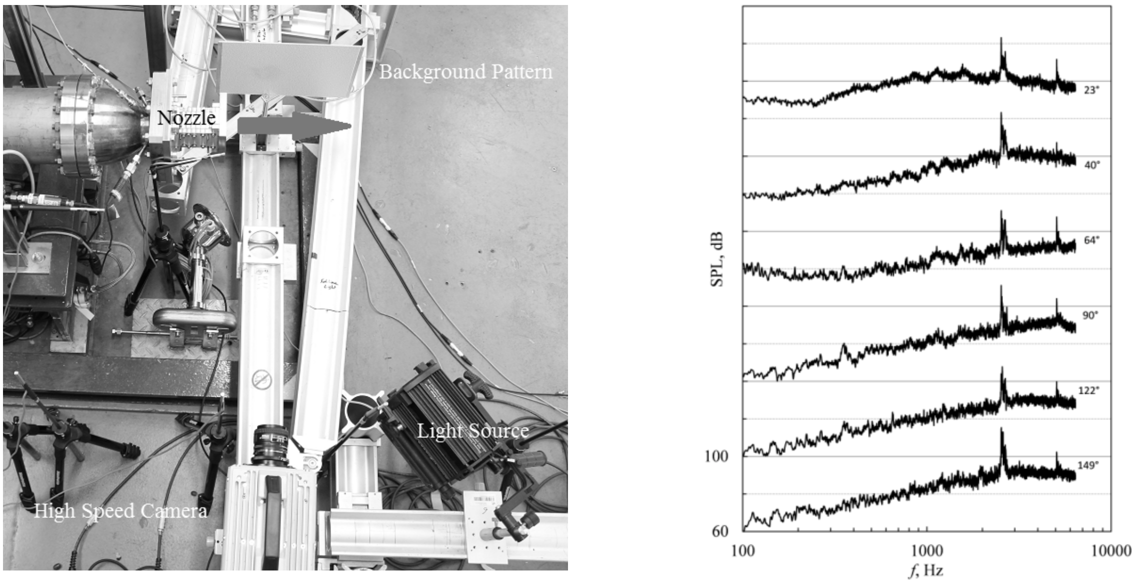


Figure 14: Test setup at M11.1 (left) and appearance of prominent tones (right)

The tests were conducted at Lampoldshausen test facilities M11.1[29] and P6.2. Test facility M11.1 features an air vitiator providing heated air at total temperature ratios up to $T_0/T_a = 5.0$ and total pressure ratios up to $p_0/p_a = 25.0$ (Fig 14, left). Depending on the operational condition, the exhaust jet produced a prominent and clearly audible tone (Fig 14, right). The prominent tones appeared in combination with heavily fluctuating shock patterns. The study was complemented by a test campaign at test facility P6.2.[27] Two cold flow subscale nozzles were tested under same operational conditions with total pressure ratios up to $p_0/p_a = 60.0$. As test specimens the TOP nozzle DLR-S1 and the TIC nozzle DLR-TIC were chosen. In opposite to the DLR-S1, the DLR-TIC displayed an additional prominent tone at 1200 Hz.

4.2 Numerical Test Bench Acoustics Investigation

The goal of this work package is to test and develop the capabilities to simulate the acoustic emission in test benches with the DLR-TAU code and the DLR-CAA solver PIANO. In a first step acoustic data from test benches using a nozzle which generates a supersonic flow with high Mach numbers as test specimen are gathered. In a second step a simulation of the experiment is performed using CAA hybrid methods, and the results are compared. The CAA method[5] applied for the simulations is a two step approach, performing first a RANS simulation to get turbulent statistic data, and using this data as input for acoustical sources in a CAA computation. For a first verification a computation was performed using experimental data[8] from the cold flow test bench P6.2 in Lampoldshausen. The flow inside a truncated ideal nozzle with design Mach number of 5 was experimentally investigated, and the simulation was conducted under the same conditions. For the test some microphones were situated to measure the acoustics. The position of the microphones is shown in Figure 15. The microphone data were acquired with 25 kHz using a Bessel filter at 8 kHz. The same data processing was applied for the numerical data. A graphical representation of the RANS and the CAA

DOI: 10.13009/EUCASS2019-515

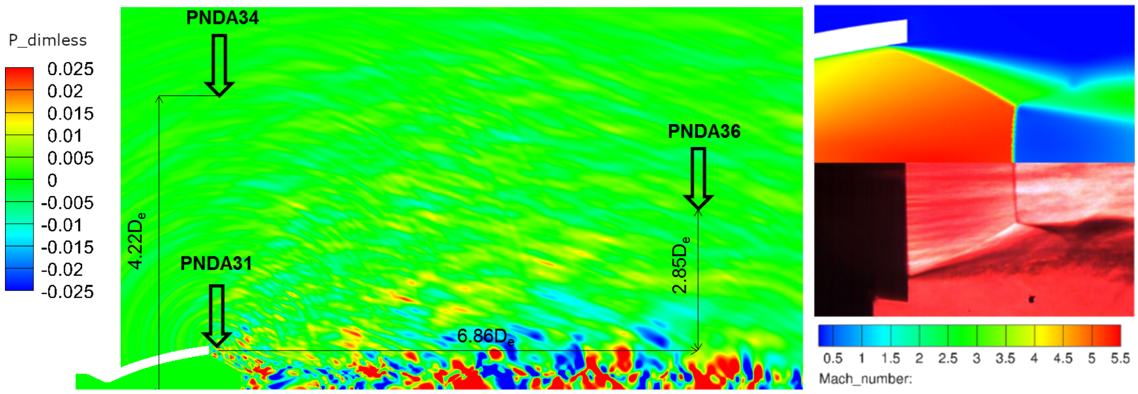


Figure 15: Right: Comparison of RANS simulation with Schlieren image. Left: Dimensionless pressure oscillations from CAA simulation and microphone positions relative to nozzle exit diameter.

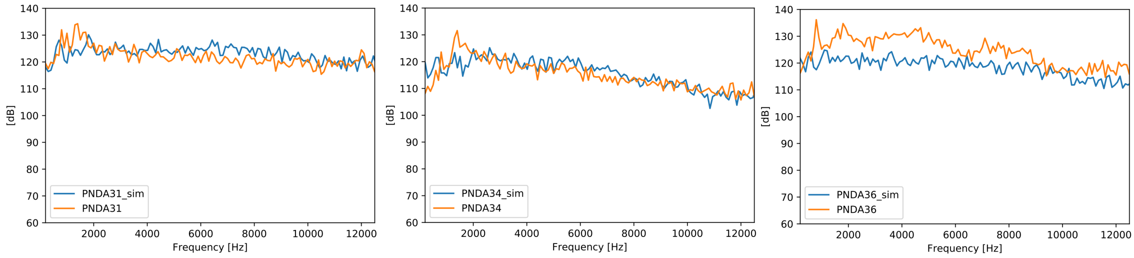


Figure 16: Evaluation of the microphone signals compared to the simulation data

simulation is shown in Figure 15. A simulation duration of 24 h was required to perform 10 ms of physical time using a 46 core machine. The 2D mesh has 359 105 grid points. This method has a great computational advantage if compared to what is necessary to perform a comparable simulation using LES or DNS. The comparison between the microphone test data and the simulation data is illustrated in Figure 16. The three microphone positions were analyzed using FFT. The results show a very good agreement for the spectra of the microphones PNDA31 and PNDA34. The results are less satisfactory for PNDA36. This behavior is in line with the underlying model of Tam and Auriault[32] which predicts good results at the sides and upstream of the noise source.

4.3 Simulation of the Vacuum Test Bench P4.1

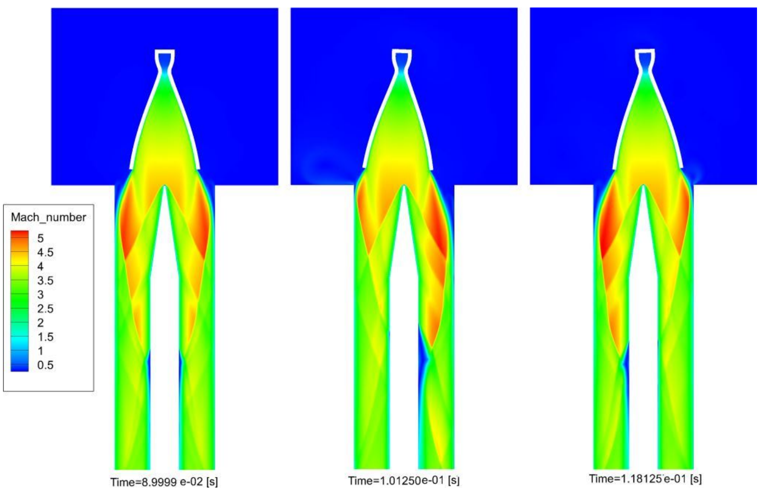


Figure 17: Instantaneous pictures of the Mach number distribution for the ungimballed postion (left), and the maximum gimballing postion (middle and right)

DOI: 10.13009/EUCASS2019-515

The goal of this work package is to simulate with the TAU code a transient gimballing simulation with the P4.1 diffuser and the VINCI engine . For this purpose TAU has an implemented method called CHIMERA which allows TAU using different overlapping meshes which can be moved during a URANS simulation. Since the CHIMERA method has not been used for such an demanding case as the P4.1 simulation a first step in order to demonstrate the capabilities of TAU to perform such simulation is needed. For this purpose a simple 2D case was defined with simple nozzle and a vacuum diffuser. This demonstration simulation case was performed using nitrogen as cold flow. In order to test the gimballing a periodic movement was defined with a period 0.045 s and maximum amplitude of 2 degrees. The simulation was initialized with no gimballing. Some instantaneous pictures of the simulation are shown in Figure 17. The calculation time needed for this simulation was 678 h on 48 core machine, being able to calculate approx. 3.21 ms simulation time each 24 h. The capability of TAU for a gimballing simulation with a supersonic jet and a vacuum diffuser has been now demonstrated for a 2D case. The next step is to set up a similar simulation for the real 3D geometry of P4.1 and VINCI. The full 3D simulation is expected to be finished by the end of 2019.

5. Thrust Chamber Wall Cooling

Cooling the thrust chamber of a liquid rocket engine is a crucial task to ensure structural integrity while reducing thermal losses to their inevitable minimum. To enable simulations of this highly interdisciplinary task and evaluate the complex interactions within the total system, a coupled simulation environment was set up. This tool chain contains the DLR-TAU code to simulate the flow of the real gas in the combustion chamber, including its chemical reactions, and the cooling channel, and a FEM simulation tool (MSC Nastran) to model the heat conduction through the solid metal parts.

Numerical simulations of the BKD combustion chamber using hydrogen as fuel and coolant and oxygen as oxidator are now being performed. The left part of Figure 18 shows the three domains of the simulation environment: the internal combustion simulation in gray, transferring heat into the metallic structure in green, which transfers it into the liquid hydrogen flowing through the cooling channel shown in orange. An example of the resulting temperature distribution of the coupled simulation in the fluid and structure is shown in the right part of Figure 18. Currently simulations of the experimental setup are carried out to gain simulation data that can be compared to existing experimental results for validation of the tool chain.

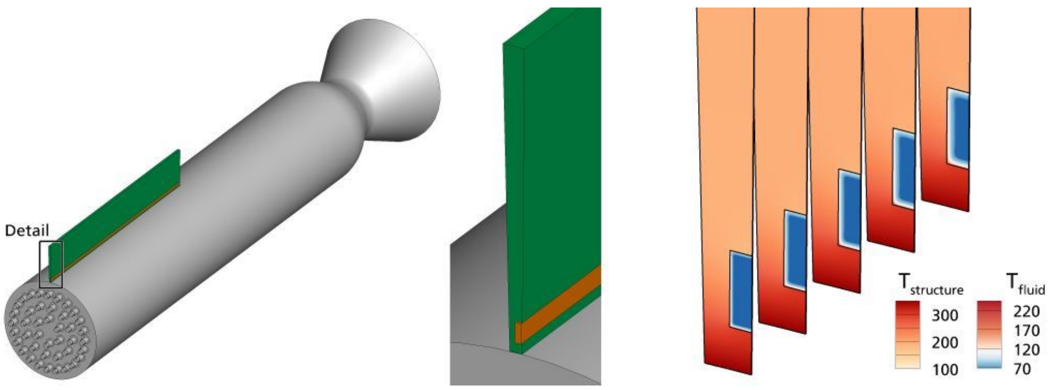


Figure 18: Solver domains for simulation of BKD (grey: combustion chamber, green: solid wall, orange: cooling channel) and cut through result of coupled simulation

To enable isolated investigations of the challenging real gas thermodynamics, a generic cooling channel has been chosen for further testing. Pressure measurements and the optical temperature distribution capturing on its outer surface allow for better comparison of the internal flow characteristics. The data gained by an experimental campaign performed with this generic geometry is useful to further assess the aforementioned simulation environment, especially with focus on the thermodynamic modeling of real gases.

6. 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.

DOI: 10.13009/EUCASS2019-515

References

[1] D. T. Banuti, V. Hannemann, K. Hannemann, and B. Weigand. An efficient multi-fluid-mixing model for real gas reacting flows in liquid propellant rocket engines. *Combustion and Flame*, 168:98–112, 2016.

[2] R. A. Baurle, G. A. Alexopoulos, and H. A. Hassan. Assumed joint probability density function approach for supersonic turbulent combustion. *Journal of Propulsion and Power*, 10(4):473–484, July 1994.

[3] E. Costa Ruiz, S. General, R. Stark, F. Strauss, and D. Schneider. 55th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit. In *Prediction of Subscale Rocket Engine Acoustic Emissions Using CAA Hybrid Methods*, Indiana, USA, 2019.

[4] E. Costa Ruiz, S. General, R. Stark, F. Strauss, and D. Schneider. 8th European Conference for Aerospace Sciences (EUCASS). In *Prediction of Subscale Rocket Engine Acoustic Emissions Using CAA Hybrid Methods*, Madrid, Spain, 2019.

[5] R. Ewert. RPM - the fast Random ParticleMesh method to realize unsteady turbulent sound sources and velocity fields for CAA applications. *13th AIAA/CEAS Aeroacoustics Conference, AIAA 2007-3506*, 2007.

[6] S. Fechter, S. Karl, V. Hannemann, and K. Hannemann. Simulation of LOx/GH2 single coaxial injector at high pressure conditions. In *53rd AIAA/SAE/ASEE Joint Propulsion Conference*, page 4765, 2017.

[7] S. Fechter, S. Karl, V. Hannemann, and K. Hannemann. Influence of non-linear mixing effects onto flow and heat transfer in rocket combustion chambers. In *53rd AIAA/SAE/ASEE Joint Propulsion Conference*, page Space Propulsion Conference, 2018.

[8] C. Genin and R. Stark. Experimental investigation of cold flow TIC nozzles, a spectral analysis. *52nd AIAA/SAE/ASEE Joint Propulsion Conference, American Institute of Aeronautics and Astronautics, 2016*.

[9] P. Gerlinger. *Numerische Verbrennungssimulation*. Springer Berlin Heidelberg, 2005.

[10] P. Gerlinger. Multi-dimensional limiting for high-order schemes including turbulence and combustion. *Journal of Computational Physics*, 231(5):2199–2228, March 2012.

[11] J. S. Hardi. *Experimental Investigation of High Frequency Combustion Instabilitiy in Cryogenic Oxygen-Hydrogen Rocket Engines*. PhD thesis, School of Mechanical Engineering, The University of Adelaide, South Australia 5005, Australia, June 2012.

[12] S. Karl, J. P. Hickey, and F. Lacombe. 31st International Symposium on Shock Waves. In *Reynolds Stress Models for Shock - Turbulence Interaction*, 2017.

[13] M. Kordula. Shock-Refracted Acoustic Wave Model for Screech Amplitude in Supersonic Jets. *AIAA Journal*, 46(3):682–689, 2008.

[14] G. Lacaze and J. C. Oefelein. A non-premixed combustion model based on flame structure analysis at supercritical pressures. *Combustion and Flame*, 159(6):2087–2103, 2012.

[15] P. C. Ma, D. T. Banuti, J.-P. Hickey, and M. Ihme. Numerical framework for transcritical real-fluid reacting flow simulations using the flamelet progress variable approach. In *55th AIAA Aerospace Sciences Meeting*, 2017.

[16] N. Perakis, O. J. Haidn, D. Eiringhaus, D. Rahn, S. Zhang, Y. Daimon, S. Karl, and T. Horchler. Qualitative and Quantitative Comparison of RANS Simulation Results for a 7-Element GOX/GCH4 Rocket Combustor. In *2018 Joint Propulsion Conference*, page 4556, 2018.

[17] H. Pitsch. *Entwicklung eines Programmpaketes zur Berechnung eindimensionaler Flammen am Beispiel einer Gegenstromdiffusionsflamme*. Diploma thesis, Rheinisch-Westfälische Technische Hochschule Aachen, 1993.

[18] B. E. Poling, J. M. Prausnitz, and J. P. O'Connel. *The properties of gases and liquids*. McGraw Hill Book Co., New York, NY, 5th edition edition, 2007.

[19] D. Schneider and C. Génin. Numerical Investigation of Flow Transition Behavior in Cold Flow Dual-Bell Rocket Nozzles. *Journal of Propulsion and Power*, 32(5):1212–1219, October 2016. doi:10.2514/1.B36010.

DOI: 10.13009/EUCASS2019-515

[20] D. Schneider, C. Génin, R. Stark, M. Oschwald, S. Karl, and V. Hannemann. Numerical Model for Nozzle Flow Application Under Liquid Oxygen/Methane Hot-Flow Conditions. *Journal of Propulsion and Power*, 34(1):221–233, January 2018. doi:10.2514/1.B36611.

[21] D. Schwamborn, T. Gerhold, and R. Heinrich. European Conference on Computational Fluid Dynamics. In *The DLR TAU-Code: Recent Applications in Research and Industry*, Delft, Netherlands, 2006.

[22] C. L. Sebourn and F. L. Shope. Research Summary on the AEDC ASTF C-2 Aeroacoustic Resonance Phenomenon. *AIAA2005-2932*, 2005.

[23] M. Seidl. *Numerische Simulation von Raketenbrennkammern mit transkritischer Einspritzung*. PhD thesis, Universität Stuttgart, 2018.

[24] M. L. Shur, P. R. Spalart, M. K. Strelets, and A. K. Travin. A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities. *International Journal of Heat and Fluid Flow*, 29(6):1638–1649, 2008.

[25] S. Silvestri, M. P. Celano, G. Schlieben, and O. J. Haidn. Characterization of a Multi-Injector GOX/CH4 Combustion Chamber. In *52nd AIAA/SAE/ASEE Joint Propulsion Conference*, page 4992, 2016.

[26] J. Smith, D. Klimenko, W. Clauss, and W. Mayer. Supercritical LOX/hydrogen rocket combustion investigations using optical diagnostics. *38th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, page 4033, 2002.

[27] R. Stark, S. General, and D. Schneider. 8th European Conference for Aerospace Sciences (EUCASS). In *Experimental Study on Jet Acoustics of Subscale Rocket Nozzles with Different Divergent Contour Design*, Madrid, Spain, 2019.

[28] R. Stark, C. Génin, D. Schneider, and C. M. Fromm. Ariane 5 Performance Optimization Using Dual-Bell Nozzle Extension. *Journal of Spacecraft and Rockets*, 53(4):743–750, July-August 2016. doi:10.2514/1.A33363.

[29] R. Stark, F. Strauss, S. General, and D. Schneider. 32nd International Symposium on Shock Waves (ISSW32). In *Experimental Study on Shock Pattern and Noise Emission of a Rocket Nozzle Exhaust Jet*, Singapore, 2019.

[30] P. Stoll. *Entwicklung eines parallelen Mehrgitterverfahrens zur Simulation der Verbrennung in kompressiblen und inkompressiblen Strömungen*. PhD thesis, Universität Stuttgart, 2001.

[31] C. Tam and L. Auriault. Jet Mixing Noise from Fine-Scale Turbulence. *AIAA Journal*, 37:145–153, 1999.

[32] C. Tam, N. Pastouchenko, and K. Viswanathan. Fine-Scale Turbulence Noise from Hot Jets. *AIAA Journal, vol. 43, pp. 1675-1683*, 2005.

[33] C. Tam, J. Seiner, and J. Yu. Proposed Relationship between Broadband Shock Associated noise and screech Tones. *Journal of Sound and Vibrations*, 110(2):309–321, 1986.

[34] C. E. Tinney, P. Panickar, and P. Vogel. Aeroacoustics of a Planar Multistream Supersonic Nozzle with Aft Deck and Sidewalls. *AIAA Journal*, 56(10):3926–3937, 2018.

[35] F. Tonti, J. Hardi, T. Horchler, S. Fechter, and M. Oschwald. Influence of numerical model setup on the response of acoustically forced LOx/H_2 flames, July 2019.

[36] F. Tonti, J. Hardi, S. Karl, and M. Oschwald. Unsteady Modelling of LOx/GH_2 Flame Response to Longitudinal Chamber Mode Forcing. In *Joint Propulsion Conference*, Cincinnati,OH, July 2018.

[37] F. Tonti, S. Karl, S. Beinke, J. Hardi, and M. Oschwald. Numeircal Modelling of Acoustic Excitation of High Frequency Combustion Instabilities in an Experimental Combustor. In *Space Propulsion Conference*, Seville, Spain, May 2018.

[38] B. Wagner, D. Schneider, S. Beinke, D. Suslov, C. Génin, J. Hardi, M. Oschwald, E. Costa Ruiz, S. Karl, S. Fechter, D. Banuti, K. Hannemann, S. Jack, M. Alder, M. Rudolph, T. Eggers, P. Gerlinger, and M. Lempke. 7th European Conference for Aerospace Sciences (EUCASS). In *Rocket Propulsion in TAU: An Overview of the DLR ProTAU Project*, Milan, Italy, 2017.

[39] V. P. Zhukov. Kinetic model of alkane oxidation at high pressure from methane to n-heptane. *Combust. Theor. Model.*, 13(3):427–442, 2009.

DOI: 10.13009/EUCASS2019-515

[40] V. P. Zhukov. The impact of methane oxidation kinetics on a rocket nozzle flow. *Acta Astronautica*, 2019.

[41] V. P. Zhukov and A. F. Kong. A Compact Reaction Mechanism of Methane Oxidation at High Pressures. *Prog. React. Kinet. Mec.*, 43(1):62–78, 2018.

[42] V. P. Zhukov, V. A. Sechenov, and A. Yu. Starikovskii. Spontaneous ignition of methane–air mixtures in a wide range of pressures. *Combust. Explo. Shock+*, 39(5):487–495, 2003.

[43] V. P. Zhukov and D. I. Suslov. Measurements and modelling of wall heat fluxes in rocket combustion chamber with porous injector head. *Aerosp. Sci. Technol.*, 48:67–74, 2016.