DOI: 10.13009/EUCASS2017-42

Rocket Propulsion in TAU: An Overview of the DLR ProTAU Project

Bernd Wagner[1]
Deutsches Zentrum für Luft- und Raumfahrt e.V. (DLR) - German Aerospace Center
Institute of Space Propulsion, Management
Lampoldshausen, 74239 Hardthausen, Germany

Dirk Schneider; Scott Beinke; Dmitry Suslov; Chloé Génin; Justin Hardi; Michael Oschwald
DLR, Institute of Space Propulsion, Rocket Propulsion Department
Lampoldshausen, 74239 Hardthausen, Germany

Eugeni Costa Ruiz
DLR, Institute of Space Propulsion, Test Facility Technology Department
Lampoldshausen, 74239 Hardthausen, Germany

Sebastian Karl; Stefan Fechter; Daniel Banuti[2]; Klaus Hannemann
DLR, Institute of Aerodynamics and Flow Technology, Spacecraft Department
Bunsenstr. 10, 37073 Göttingen, Germany

Sebastian Jack; Marko Alder; Martin Rudolph[3]; Thino Eggers
DLR, Institute of Aerodynamics and Flow Technology, Spacecraft Department
Lilienthalplatz 7, 38108 Braunschweig, Germany

Peter Gerlinger; Markus Lempke[4]
DLR, Institute of Combustion Technology, Numerical Simulation Department
Pfaffenwaldring 38-40, 70569 Stuttgart, Germany

Abstract

The ProTAU project is an internal cooperation between several DLR institutes to advance the independent DLR expertise in modelling rocket propulsion systems and their components. The focus is on the development and validation of the DLR in-house CFD code TAU. The project began in early 2014 after an initial two-year phase of collaborative work between multiple DLR institutes. The project unifies the complementary code development, validation, simulation, and experimental work conducted by the various institutes. The collaboration facilitates a continuous exchange between developers and end-users allowing developers to respond more rapidly to user requirements and a more efficient development progress.

In this paper, a brief overview of the ProTAU project structure is given. ProTAU is organized into four research topics: combustion chamber, nozzles, test facilities, and cooling. Highlights of the progress achieved will be shown for each main work package.

---

[1] Corresponding author: Bernd.Wagner@dlr.de

[2] Former employee of the Spacecraft Department - Göttingen
[3] Former employee of the Spacecraft Department - Braunschweig
[4] Former employee of the Numerical Simulation Department

1. Introduction

The aim of the project ProTAU is the further development and qualification of the DLR-TAU code as one further step towards a virtual rocket engine. In order to achieve this goal, the already started establishment of an independent DLR competence for the modelling and calculation of rocket propulsion systems and their components is continued and intensified in ProTAU. The project brings together expertise from multiple DLR institutes including the Institute of Space Propulsion (RA), the Institute of Aerodynamics and Flow Technology (AS) and the Institute of Combustion Technology (VT).

Within the scope of ProTAU, the DLR-TAU code with its space extensions is further developed and validated for specific applications in the field of space propulsion. Thus, the space extension of TAU is the focus of the development part of the project. The institutes AS, RA and VT work closely together in the planning and implementation of the project. The various competences involved in the development and application of TAU in the area of space propulsion (AS), the use and further development of special numerical methods for combustion processes (VT) and the operation and development of test stands, research engines and their components (RA) are combined. In addition to the further development and validation of the DLR-TAU code, this collaboration will lead to the flow-mechanical design of future vehicle systems as well as improved support for experimental work on rocket engines and the use of these results.

2. Description of the ProTAU Project

The project is subdivided into four sections: combustion chamber, nozzles, test facilities and cooling channels. A schematic view of ProTAU’s way towards an independent numerical expertise can be found in Figure 1. In the first project phase of 3 years, the topics have been treated independently. Nevertheless, similarities have been used and a strong cooperation and exchange takes place not only within the main topics but also between them. For example the real gas library, developed for the combustion chamber process, has also been used for simulation in cooling channels. Findings of fluid structure interaction were exchanged between nozzle and cooling channels. For the future, the interaction will be enhanced, while the physical problems in a rocket engine are highly coupled.

The development of the DLR-TAU code for the calculation of flow and combustion processes in cryogenic hydrogen / oxygen rocket engines is the focus of the combustion chamber. Suitable models for the treatment of supercritical and subcritical injection and combustion are implemented and validated.
The validation and evaluation of the accuracy of the newly implemented methods within ProTAU is based on experimental and numerical reference data. For this purpose, test campaigns are carried out on specific test devices (model nozzles, cooling channels with supercritical flow, high-pressure combustion chambers for optical flow and combustion diagnostics) for the generation of suitable validation data. Furthermore, reference solutions for simplified model problems are generated using highly accurate numerical simulation methods.
The application of the DLR-TAU code to model combustion chambers with a cryogenic fuel combination for the investigation of HF instabilities is also carried out in the combustion chamber work package. The knowledge and predictability of these phenomena is an important requirement for the design of future combustion chamber technologies.

In the section nozzle, numerical and experimental investigations on deformed nozzles are closely coupled. The improved understanding of the separation behaviour at high external pressure (start, sea level) is important for the use of high expansion thrust nozzles, which directly results in an increase in the performance of the entire space transportation system. Furthermore, the operating behaviour of dual-bell nozzles as a future concept for altitude-adaptive nozzles is investigated and evaluated by coupled numerical and experimental analyses.

The test facilities section covers the qualification and further development of the TAU process for the design and optimization of test stands. Firstly, suitable numerical modelling strategies (turbulence models) for flow phenomena in cold gas test stands are identified. The results obtained enable an improved understanding of the operating behaviour of the test facilities, e.g. occurrence of ejector pumping under certain operating conditions, and an evaluation of the precision of the applied calculation method (TAU). Furthermore, the implementation and validation of suitable numerical models for the treatment of water sprays in hot gas test facilities is carried out. A coupling of TAU with the spray simulation method SPRAYSIM (VT) is implemented. The numerical results are evaluated by means of test data obtained from water cooled hot-gas test facilities.

DOI: 10.13009/EUCASS2017-42

In order to significantly increase the efficiency, power and specific impulse of future rocket engines, high combustion chamber pressures are required above all. If the combustion chamber pressure is increased in order to increase the power of the engine, the energy density and the thermomechanical loading of the combustion chamber structure are also increased. Because most of the components are already up to the material limit, a serious improvement in cooling and material behaviour is an important criterion for the further development of space transportation systems. Therefore the multidisciplinary analysis of flow and heat transport processes in cryogenic cooling channels is in the focus of the sub-section cooling channels. First, the turbulence and real gas models present in TAU are evaluated and possibly improved for the calculation of flows in cooling channels. The numerical work is supported with validation data obtained by conducting measurement campaigns on one-sided heated test cooling channels.

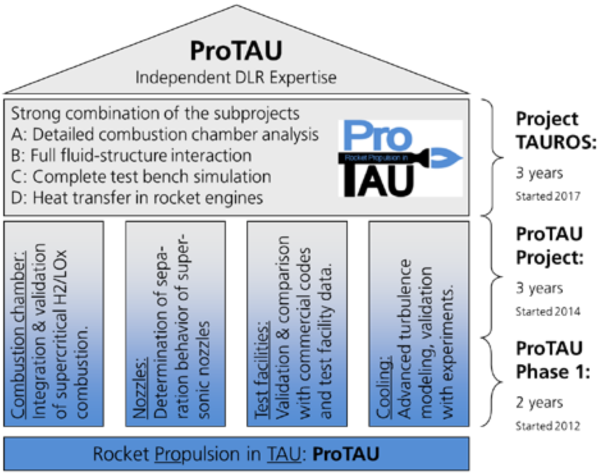

Figure 1: ProTAU at a glance [30].

The project ProTAU is a milestone for the development of the virtual rocket engine and paves the way for further detailed projects at DLR.

3. Overview of Key Results of the ProTAU Project

In the following section a brief overview of key results obtained during the ProTAU project are presented. Due to the limitations of this format, it should be referred to the publications of the dedicated topics. Some of them, but not exhaustive, have been mentioned in the text and can be found in the reference list. The methodology and the results are described in detail in these references.

3.1. Combustion Chamber

The combustion chamber comprises numerous physical problems, like propellant spray, combustion, multiphase, turbulent interaction, instability, just to name some. Each of these problems is huge challenge for numerical modelling, is subject of intensive research, and would be worth of entire projects. Due to understandable limits in resources ProTAU found a good compromise between developing a reliable and accurate tool and covering all topics with a reasonable effort. Therefore, it is very important that the development of the rocket propulsion extension of the TAU solver is accompanied by the unique and good experiments with BKC and the high order simulations of rocket combustors.

3.1.1. Real TAU Development and Application

The focus of this part was on the extension of the TAU solver to handle cryogenic combustion processes in rocket combustion chambers. The first part was on the development of an extension for the flow solver to efficiently handle cryogenic fluids that are present in rocket combustion chambers, e.g. liquid oxygen. For that scope the Multi-Fluid-Mixing model was developed in [2][4][6] that allows the fluid description of one non-ideal gas component in the limit of stationary hydrogen-oxygen diffusion flames. The fluid behavior of the non-ideal component is described by the modified Benedict–Webb–Rubin equation of state (EOS), a highly accurate description for the fluid behavior. All other components are described with the ideal-gas law. It is justified as long as the fluid components mix solely in a region there the ideal gas mixing law can be applied, e.g. in the combustion zone or at non-cryogenic temperatures.

For steady-state hydrogen-oxygen flames this approach was justified based on LES and DNS investigations of Oefelein [22] as well as Lacaze and Oefelein [19]. In this case the flame acts as barrier that prevents the direct mixture of hydrogen and oxygen.

The Multi-Fluid mixture model description facilitates the numerical treatment of mixtures within the flow solver. Only for the handling of one non-ideal fluid component, the TAU flow solver has to be adapted. This was done by a two-dimensional tabulation method that allows for a computationally efficient integration of the non-ideal EOS component [3]. This allows the numerical simulation of the cryogenic fluid behavior at little additional costs compared to the ideal gas description. For that case an equidistant tabulation method was chosen that covers the whole density and temperature interval used in the simulation.

The sub-critical thermodynamic modeling is based on a simple mixture model in thermodynamic equilibrium of the non-ideal fluid component. Based on that assumption the average behavior vapor and liquid can be described using a thermodynamic equilibrium approach (constant pressure and temperature within spinodal EOS region). This approach is justified as long as the thermodynamic processes are faster compared to the numerically resolved time scales. This simple model allows the resolution of subcritical phase transition effects without the need to track the interface explicitly in the numerical framework.

The numerical approach based on the extended TAU framework has been validated with the Mascotte A60 test case [4] as well as the experiments in the BKC campaign (see section 3.1.4). In Figure 2 a CFD simulation for the BKC test case 1 at supercritical conditions is shown, more details can be found in ref. [11]. At these conditions the comparison of the TAU flow solver reproduces the experimental measurements available in literature.

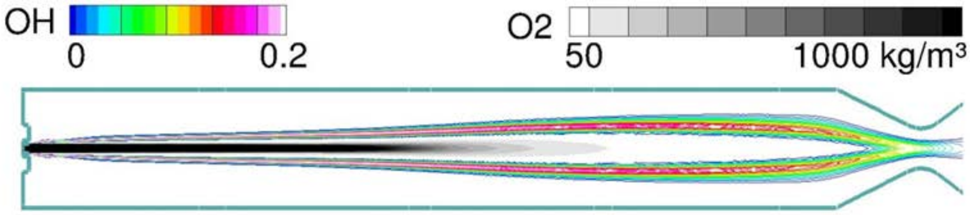

Figure 2: Numerical result for the liquid oxygen core and OH*-emission for the BKC load step1 at supercritical conditions.

3.1.2. High Order Methods

In many rocket combustion chamber simulations the numerical results improve if time accurate simulations are performed. This is due to unsteady effects which are often present in rocket combustors. Moreover, time accuracy is always required if combustor oscillations have to be resolved [26]. In such cases it is advantageous to use high order spatial and temporal discretization techniques and URANS or LES for the turbulent flow field. Corresponding simulations give insight into the physical and chemical processes and allow the validation and development of less detailed modelling approaches.

In the present case a high order MLPld (multi-dimensional limiting process [32], low diffusion [16]) scheme is used to achieve spatial discretization up to sixth order. MLP uses information from diagonal volumes of a discretization stencil and thus combines the different coordinate directions. It interacts with the TVD limiter in such a way, that local extrema at corner points of a volume are avoided. In this way a good shock resolution is achieved even if the shock is oblique to the computational grid. Moreover, convergence is improved compared to conventional TVD (total variation diminishing) limiters [16]. While the additional numerical effort for the high order discretization is very low, accuracy improves significantly [27][20] compared to conventional second order techniques. For discretization in time a 2nd or 3rd order BDF (backward differentiation formula) method is incorporated into an implicit LU-SGS (Lower-Upper symmetric Gauss-Seidl) scheme. With respect to turbulence both URANS and DDES (delayed detached eddy) simulations are performed. Finite-rate chemistry describes combustion and for turbulence chemistry interaction either a multi-variate assumed PDF (probability density function) closure or a Lagrangian transported PDF particle solver is used.

The described techniques are used to simulate the single injector PennState model rocket combustor [21]. Figure 3 shows a contour plot of the instantaneous temperature distribution obtained by a DDES. Results of different high order MLP URANS simulations may be found in [20], 4th order result for a TPDF simulation in Ref. [15]. Close to the LOX post, the flow field is highly unsteady and there are strong chemical non equilibrium effects.

DOI: 10.13009/EUCASS2017-42

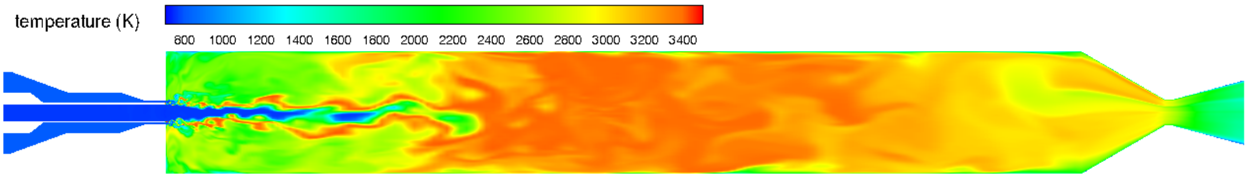

Figure 3: Calculated instantaneous temperature distribution of the PennState combustor from a 5th order DDES with multi-variate assumed PDF closure and finite-rate chemistry.

3.1.3. Validation Data from BKC Campaign

In the framework of the DLR-project ProTAU, we have performed tests to create an extended data base for numerical tool validation for high pressure LOX/H2 combustion.
During the experimental investigations a windowed DLR subscale thrust chamber model “C” (designated BKC) has been operated over a broad range of conditions at reduced pressures of approximately 0.8 (4 MPa), 1 (5 MPa) and 1.2 (6 MPa) with respect to the thermodynamic critical pressure of oxygen. Liquid oxygen and gaseous hydrogen have been injected through a single coaxial injector element at temperatures of ca. 120 K and ca. 130 K, respectively.

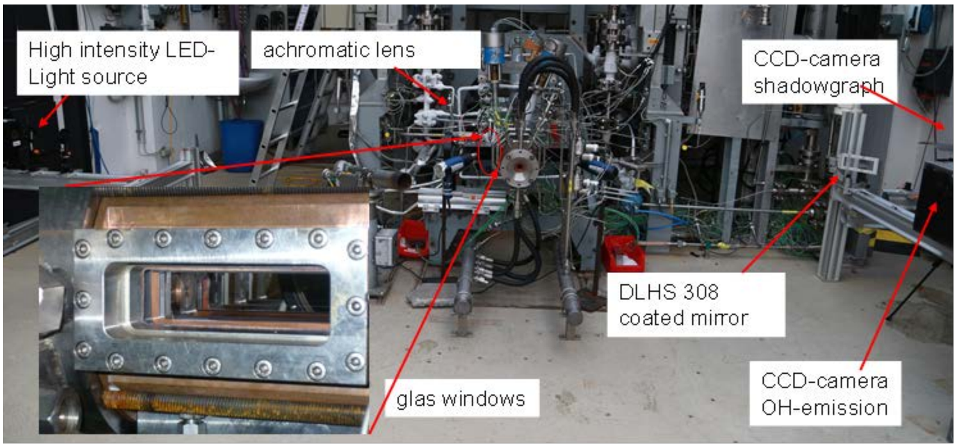

Figure 4: DLR subscale combustion chamber model “C” (BKC) with optical access at European technology test bench P8

Application of high-speed, time-resolved optical diagnostics in combination with conventional measurement techniques has provided quantitative and qualitative evidence of dissimilar combustion behaviour at various reduced pressure levels through visualisation and analysis of flow-field data. Such observations and results highlight the importance of further experimental investigations to provide additional fundamental information and enhance the current understanding of high-pressure, liquid rocket engine combustion. The data are expected to be of value in the validation of numerical tools for the design of liquid rocket engine thrust chambers [28][29] and they have intensively been used in ProTAU, see ref. [11][26].

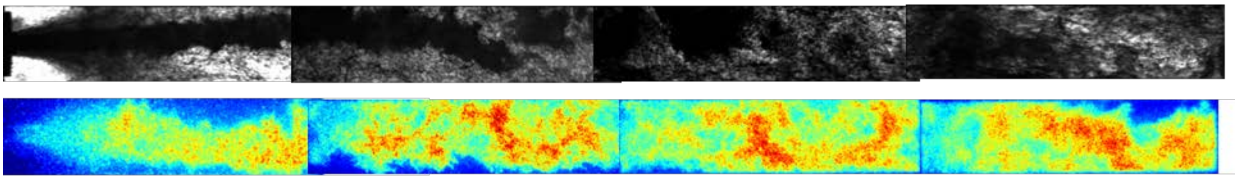

Figure 5: Combined shadowgraph and OH images at combustion chamber pressure 4.0 MPa and mixture ratio 4

3.1.4. CFD modelling of Multi-element Combustion Chamber BKH

The BKH experiments conducted at the DLR Institute of Space Propulsion are used to investigate high frequency combustion instability phenomena. A diagram showing the experimental setup of BKH is shown in Figure 6. In the experiments a series of coaxial oxygen-hydrogen injection elements are subjected to an imposed acoustic disturbance representative of combustion instability phenomenon. The observed flame response to the imposed disturbance is

DOI: 10.13009/EUCASS2017-42

analysed to investigate the effect of acoustic excitation on jet breakup and combustion processes at various operating conditions. CFD modelling of the BKH experiments was pursued in the scope of the ProTAU project to complement and provide further insight into the experimental results. The necessary capabilities needed to model the BKH experiments were developed via collaboration with developers from the Institute of Aerodynamics and Flow Technology. This included the capability to model the trans-critical injection and combustion processes in BKH, and methods for imposing representative acoustic disturbance upon unsteady models of BKH injection elements.

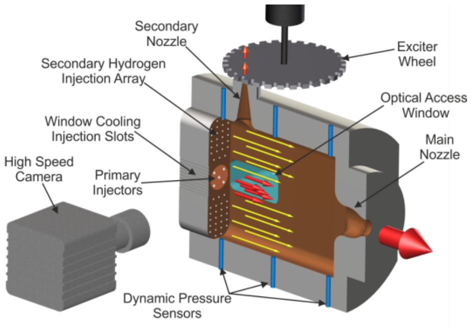

Figure 6: Concept diagram of BKH

A steady-state model of an unexcited BKH experiment was computed [8]. This was the first model of a multi-element combustion chamber operating with cryogenic liquid propellants computed with the TAU code. The results were compared with available optical images of the primary flame zone from BKH experiments. Good experimental agreement was obtained with the model results reproducing the length of the LOx core and primary flame distribution observed experimentally. The model of the BKH chamber also facilitated participation in the 3rd Rocket Engine Stability iniTiative (REST) modelling workshop [9].

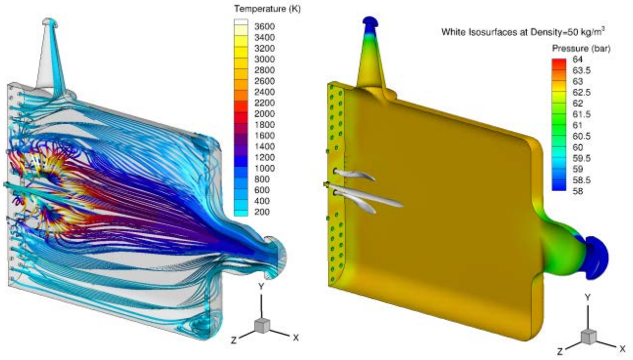

Figure 7: Results of the BKH steady-state model

The TAU code was also applied to model the unsteady response of BKH experiments to acoustic excitation. Modelling of BKH cold flow experiments were used study the BKH acoustic excitation system and how to reproduce the acoustic disturbance numerically [10]. Other methods for acoustically exciting a representative sub-domain containing single or multiple BKH injection elements were later developed and used to investigate the unsteady flame response to longitudinal and transverse acoustic disturbances [7]. Models of single BKH injection elements have been used to investigate the flame response to acoustic excitation in detail. The single injector models have been shown to reproduce key experimental observations such as the flattening and retraction of the flame when subjected to a transverse acoustic velocity disturbance. The resolved numerical results provide further insight and details of the resulting flame response than what is possible experimentally and will be employed in future work to investigate the flame response at different excitation and operating conditions.

3.2. Supersonic Rocket Engine Nozzles

3.2.1. Dual Bell Nozzles

During the project ProTAU a numerical model is developed, to predict the dual-bell transition behaviour under cold flow conditions. Validation data for the numerical model are obtained by several test campaigns conducted at DLR's cold flow test facility P6.2 in Lampoldshausen. Detailed numerical parameter studies are carried out, to find the best numerical setup in terms of turbulence modelling, physical time step, feeding pressure gradient, etc. for the dual-bell application [25]. Figure 8 and Figure 9 illustrate the validation of the numerical 2D results against the experimental data, for the dual-bell transition to altitude mode and the hysteresis behaviour.

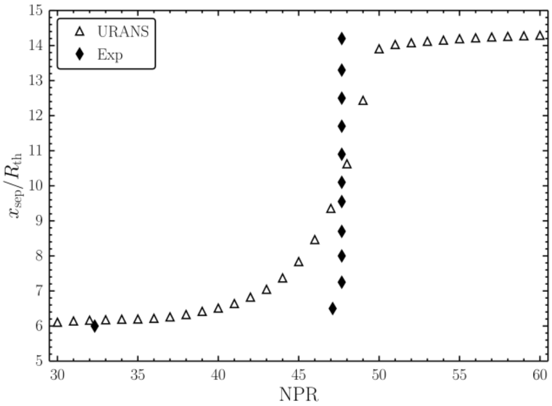

Figure 8: Separation position dependence on the nozzle pressure ratio during transition to altitude mode

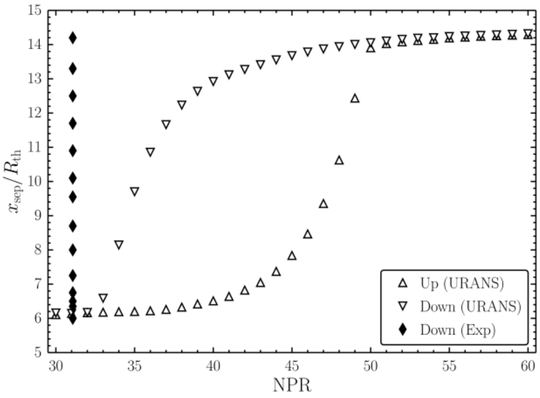

Figure 9: Hysteresis behaviour of the investigated dual-bell nozzle

It can be observed, that the developed numerical model predicts the transition and hysteresis of the investigated dual-bell nozzle model with sufficient accuracy.

In addition to the 2D axisymmetric simulations full 3D simulations are conducted, in order to investigate the influence of 3D effects on the dual-bell transition behaviour. Figure 10 depicts the Mach number distribution for a full 3D simulation of the dual-bell nozzle in sea level mode. The comparison of the 2D and 3D results yields almost no difference for the dual-bell transition behaviour.

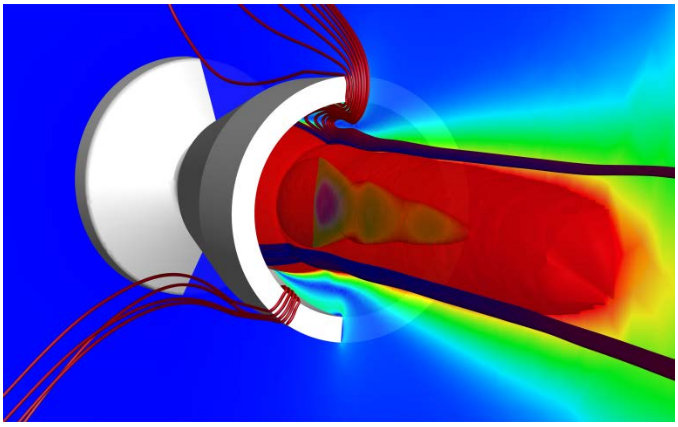

Figure 10: Mach number distribution of the dual-bell nozzle in sea level mode

In the end of the project the developed numerical model is applied on stiff ovalized dual-bell nozzles, in order to investigate the impact of a contour ovalization on the dual-bell transition behaviour. Therefore, the transient transition of different ovalized dual-bell nozzles is simulated. The simulations yield a clear impact of the contour ovalization on the dual-bell transition and the hysteresis behaviour.

3.2.2. Ovalized Nozzles

During the transient start-up and shutdown process of a rocket engine high side loads occur, due to unsymmetrical flow patterns. The resulting deformation, and its retroactive effect onto the internal flow, excite the nozzle structure and can lead to its fatal damage. Flow separation at the nozzle wall amplifies the deformation [12]. The aim of this work package of the ProTAU project was to develop a validated method to simulate these retroactive effects.

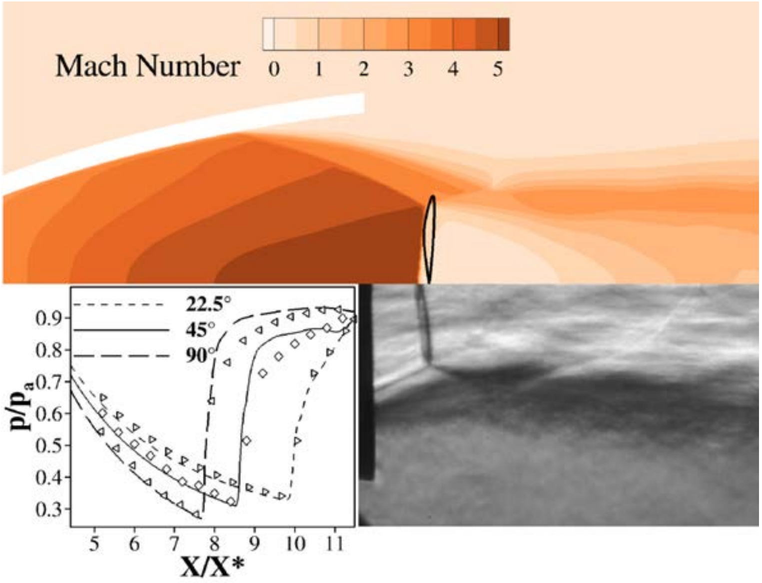

Figure 11: Typical flow patterns in a deformed nozzle, numerical data (upper part) and experimental Schlieren image (lower part), together with wall pressure distributions

Since the investigation of deformed nozzles is very challenging, both for the numerical as well as the experimental setup, the complexity of the deformation has therefore been increased step by step to be able to investigate the underlying phenomena. In each phase –rotation-symmetric and flow response to a defined deformation - numerical and experimental studies were carried out to collect validated data [14][17]. For further studies a fully coupled fluid-structure simulation environment has been set up and extended for the specific requirements of nozzle simulations. The numerical simulations show that the TAU code is well capable to predict the flow patterns in a deformed cold gas rocket nozzle. While preliminary studies with the Spalart-Allmaras turbulence model were able to predict the correct qualitative influence of the deformation parameters onto the flow patterns in a quick and robust way, the more sophisticated Reynolds-stress turbulence model yielded much better results especially when focussing onto the flow phenomena in the recirculation area [12][18]. All numerical results have been validated with experimental Schlieren images and wall pressure data measured at the P6.2 test bench at DLR Lampoldshausen [13]. Figure 11 shows the typical flow patterns in a deformed nozzle. The upper half shows the Mach number distribution and the triple line, i.e. Mach disk border, obtained from the numerical simulation at a cutting angle of 45°. Additional to the corresponding experimental Schlieren image, the lower half shows the comparison of wall pressure distributions along three cutting angles. Numerical and experimental data show good overall agreement.

3.3. Test Facilities for Rocket Engine Tests

3.3.1. Cold Flow Facilities

One goal of the ProTAU project was to assess the applicability and accuracy of the TAU solver for flows in complex test stands. The DLR P6.2 high altitude test stand was selected as a representative configuration and detailed numerical analyses of different operating conditions were carried out. The main focus was on the assessment of the performance of different RANS turbulence models ranging from one-equation to Reynolds Stress models. The considered test stand features a closed test chamber and different ejector pumps to achieve high altitude simulation through controllable ambient pressures in the test chamber. The main parameter used to assess the quality of the numerical results was the test chamber pressure and its temporal evolution during unsteady operating conditions. A schematic of the test stand layout including an exemplary result for the flow field and a comparison of numerical and experimental test chamber pressures for steady operating conditions are shown in Figure 12.

The complex flow fields in test stands impose a challenging problem for RANS based CFD analyses. Flow separation and reattachment occurs in the diffusor and ejector. This includes shock boundary layer interaction, the presence of shock trains and mixing and entrainment of exhaust gas from the test nozzle and the ejector pump. Nevertheless, good agreement between the experimental and numerical test chamber pressure over the entire range of operating conditions was achieved with Reynolds Stress turbulence modelling. The maximum deviation was 15 %. A particular success was the identification of the driving mechanisms of test stand pumping which occurs for certain ejector configurations by means of unsteady CFD simulation. A periodic change between fully separated and partially attached supersonic flow in the ejector is responsible for an alternating fluid mechanical coupling and decoupling of the test chamber with the ambient state downstream of the ejector exit.

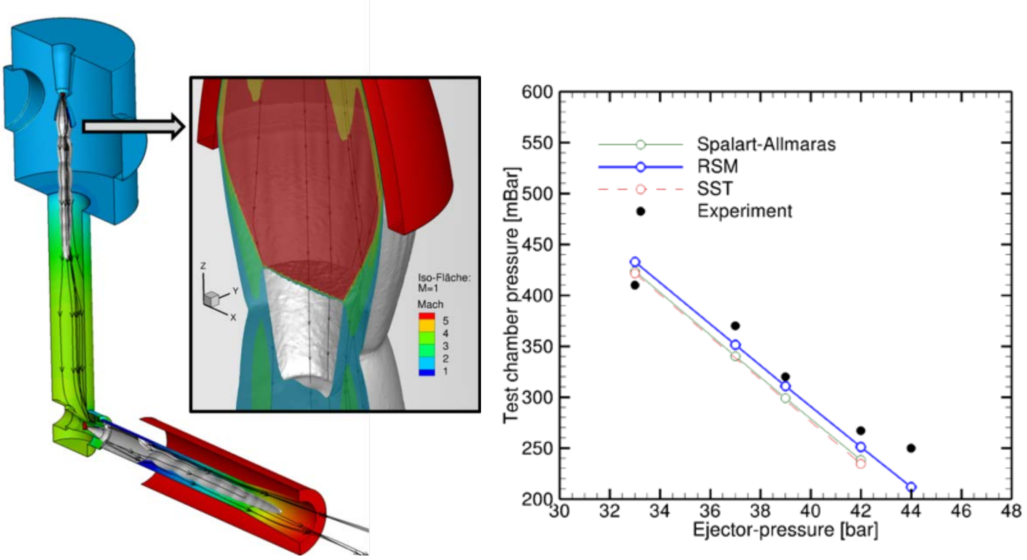

Figure 12: P6.2 flow field (left) and test chamber pressures (right).

3.3.2. Hot-gas Test Facilities

The primary goal of the work related to hot gas test facilities was to establish a coupling procedure between DLR’s Lagrangian Spray Modelling Software SPRAYSIM and the Tau CFD solver and to validate the coupled analysis procedure against canonical test cases and experimental results from existing test facility configurations. Tests of a Methane / Oxygen thrust chamber without turbopumps at the P3.2 test stand were selected for demonstration of applicability of the combined Lagrangian Spray and CFD simulation methodology. Preparatory studies identified a suitable reaction mechanism to account for the chemical non-equilibrium effects in the exhaust plume. A schematic of the computational domain comprising the engine exit, diffusor tube and spray injector including an exemplary result for the flow field and a comparison of numerical and experimental diffusor wall temperatures are shown in Figure 13. In this figure, pressure is shown as grey-scale, water droplet trajectories are indicated by red lines, streamlines are shown in black and the blue regions correspond to intense vaporization of the water spray. The mass fluxes of the engine exhaust and the water spray were 100 kg/s and 440 kg/s, respectively. The right part of Figure 13 shows good comparison between the numerical prediction and the experimentally observed wall temperatures in the diffusor tube for the results which include the water spray. About 30% of the injected water droplets evaporate inside the diffusor tube.

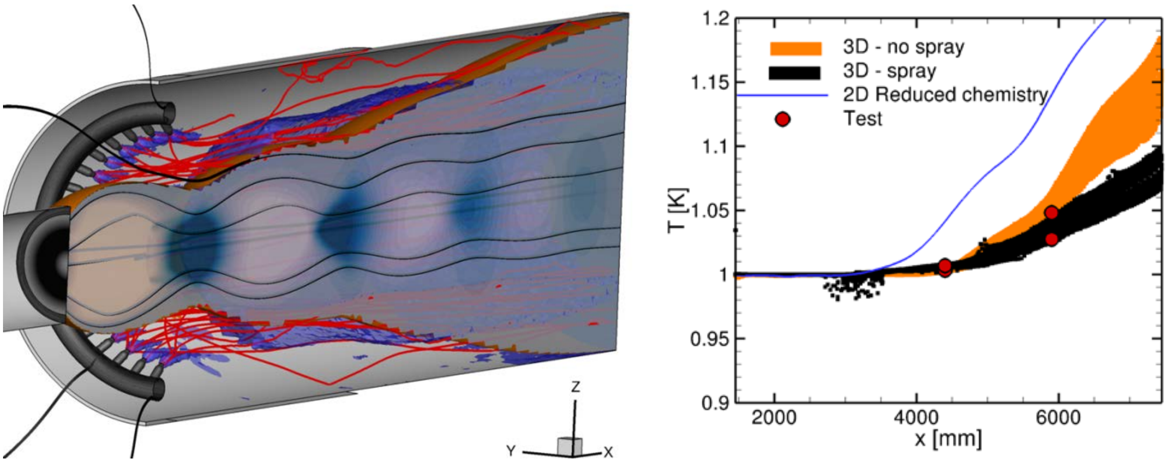

Figure 13: Flow field in the test stand diffusor tube (left) and diffusor wall temperatures for different modelling assumptions (right).

DOI: 10.13009/EUCASS2017-42

3.4. Cooling Channels

3.4.1. Numerical Activities

The numerical studies investigate real propellant high aspect ratio cooling channel flows by means of multidisciplinary coupling of CFD and a finite element method. Various modifications of the DLR-TAU code were necessary in order to sufficiently predict the thermodynamic properties of the fluid at near- and supercritical conditions. These accurate thermodynamic properties are stored in a look-up table that allows an efficient usage within the CFD code [5]. Further modifications of the CFD code include the possibility to define a heat flux distribution as boundary condition.

A first numerical verification is based on a simple methane pipe flow subjected to a constant wall heat flux by Wang [31]. Because of the stiff thermodynamics for real propellants and security restrictions for experiments, nitrogen has been chosen for the sake of a qualitative comparison with literature. Noteworthy is that a drop of the convective heat transfer into the fluid occurred for a critical heat flux value, similar to the methane investigations [31]. Detailed discussions of the results can be found in [24]. Further test cases, such as results from Pizzarelli [23], are used to verify the correct implementation of the aforementioned modifications.

The results reveal that the DLR-TAU code is capable of predicting trans- and supercritical fluids with sufficient accuracy. However, further work is necessary with respect to an improvement in numerical stability and a validation for different gases based on experimental measurements. Finally, a partitioned solution of the coupled fluid-structure system based on TauPython [1] will be established in the TAUROS project.

3.4.2. Cooling Channel Facility M51.3

Today rocket engines using high-energy cryogenic propellants play a major role due to the high combustion enthalpy and the high specific impulse of these propellants. High temperature differences between the hot combustor gases and the cooling fluid in combination with high heat transfer coefficients yield to extreme heat flux levels through the combustion chamber wall. The reliable operation of rocket combustion chambers at such high thermal and mechanical loads is achieved with highly efficient cooling. For optimal cooling design with minimal hydro-dynamic losses the precise knowledge of heat transfer processes in rocket engines is important.

Optimization of the heat transfer management is a key issue in designing a rocket combustion chamber. Therefore heat transport processes have been an ongoing interest at the DLR. Despite substantial progress in numerical simulations, needs on realistic experimental data at representative conditions both at hot gas and at coolant side for verification and development of numerical design tools still exist. In the frame of the ProTAU work package *Cooling Channel*, an experimental setup has been development and build as new test facility M51.3 as well as the test specimen EHT-LN2 for an investigation of the heat transfer processes in cryogenic fluids at high thermal loads. Application of IR-diagnostics in combination with conventional measurement techniques has provided quantitative and qualitative evidence of dissimilar heat transfer behaviour at various reduced pressure and heat load levels through visualization and analysis of experimental data. Such observations and results highlight the importance of further experimental investigations to provide additional fundamental information and enhance the current understanding of regenerative cooling in a liquid rocket engine. The data are expected to be of value in the validation of numerical tools for the design of liquid rocket engine thrust chambers.

DOI: 10.13009/EUCASS2017-42

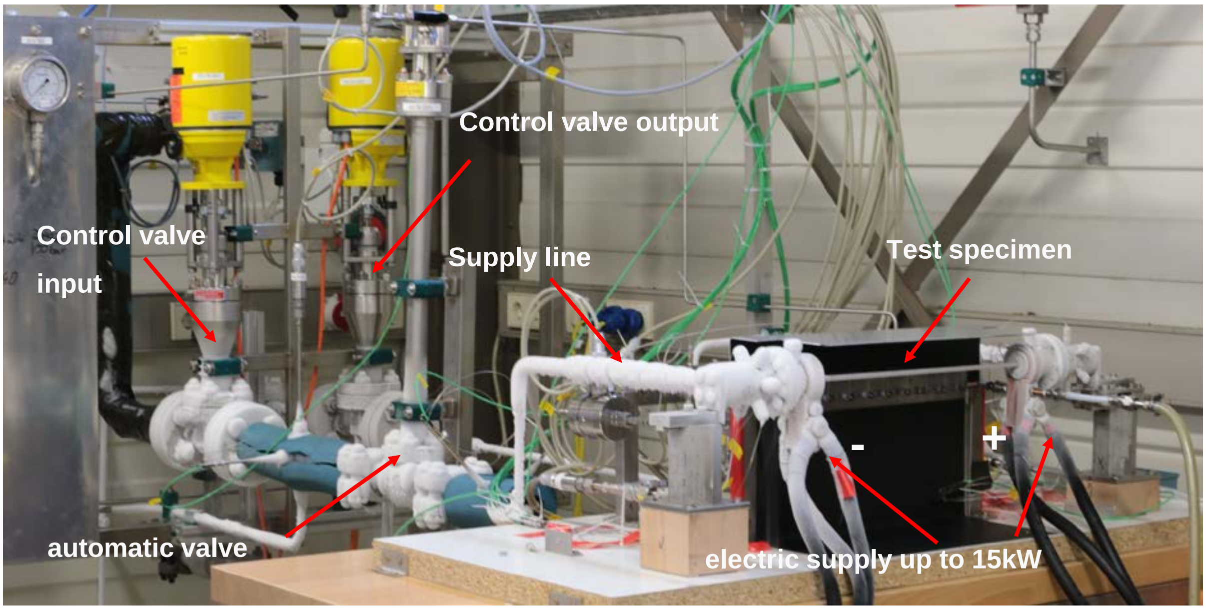

Figure 14: New DLR test facility for investigation of the heat transfer processes in cooling channels at sub- and supercritical conditions

4. Outlook TAUROS

The Project ProTAU provided the fundamentals to offer DLR's flow solver TAU the capability for rocket thrust chamber simulations. The flow solver TAU was expended with several routines and numerical tools, which have been validated against generic test cases. The aim of the project TAUROS is the utilization of the developed methods on close to real application cases. Furthermore, a coupling of the in ProTAU discrete regarded subjects like combustion chamber, nozzle, test bench and cooling channels is conducted. For example, a simulation of the combustion processes in a 42 coaxial injector combustion chamber is carried out. The obtained results for the reactive hot gas flow are used as boundary condition for cooling channel investigations. Thus, the subsystems combustion chamber simulation and cooling channel simulation are merged to an interdisciplinary overall system.

Acknowledgements

The project ProTAU as well as the successive project TAUROS are fully funded by DLR. The support of DLR space flight programmatic board is kindly acknowledged. The authors are also grateful for the additional support of Institute of Space Propulsion for the BKC test campaign and the construction of the M51.3 test facility.

References

[1] Alder, M. 2015. Development and Validation of a Fluid-Structure Solver for Transonic Panel Flutter. *AIAA Journal*. 53(12). 3509–3521.
[2] Banuti, D., K. Hannemann. 2014. Efficient multiphase rocket propellant injection model with high quality equation of state. In *Space Propulsion Conference*.
[3] Banuti, D., K. Hannemann. 2014 Application of a Real-Gas-Library Multi-Fluid-Mixing Model to Supercritical Single Injector Flow. In *50th AIAA Joint Propulsion Conference*.
[4] Banuti, D. 2015. Thermodynamic Analysis and Numerical Modeling of Supercritical Injection. PhD Thesis. Universität Stuttgart
[5] Banuti, D., K. Hannemann. 2013. Real Gas Library in Continuous Phase Propellant Injection Model for Liquid Rocket Engines. In *49th AIAA Joint Propulsion Conference* .
[6] Banuti, D., V. Hannemann, K. Hannemann, B. Weigand. 2016. An efficient multi-fluid-mixing model for real gas reacting flows in liquid propellant rocket engines. *Combustion and Flame*. 168. 98–112.
[7] Beinke, S., D. Banuti, J. Hardi, M. Oschwald, B. Dally. 2016. Modelling of a Coaxial LOx/GH2 Injection Element Under High Frequency Acoustic Disturbances. *Progress in Flight Physics*. 8.

[8] Beinke, S., J. Hardi, M. Oschwald, B. Dally. 2014. Modelling an Acoustically Perturbed Rocket Engine Combustion Chamber with Cryogenic Propellant Injection. In *19th Australasian Fluid Mechanics Conference*.

[9] Beinke, S. et al. 2014. HF-6 Test case: LOX/H2-Combustion under forced acoustic excitation. In *Proceedings of the 3rd REST Modelling Workshop*.

[10] Beinke, S., J. Hardi, M. Oschwald, B. Dally. 2013. Modelling Acoustic Excitation of High Frequency Combustion Instability Experiments. In *5th European Conference for Aeronautics and Space Sciences (EUCASS).*

[11] Fechter, S., K. Hannemann. 2017. Simulation of LOx/GH2 single coaxial injector at high pressure conditions. In *53rd AIAA Joint Propulsion Conference.*

[12] Génin, C., S. Jack, R. Stark. 2015. Flow visualization in out-of-round rocket nozzles. In *30th International Symposium on Shock Waves.*

[13] Génin, C., S. Jack. 2015 Flow separation study in stiff ovalized rocket nozzles, Part I: Experimental approach. In *51st AIAA Joint Propulsion Conference*.

[14] Génin, C., R. Stark, S. Jack. 2015. Flow separation in out-of-round nozzles, a numerical and experimental study. *Progress in Flight Physics*. 7. 269–282

[15] Gerlinger, P. 2017. Lagrangian Transported MDF Methods for Compressible High Speed Flows. *Journal of Computational Physics.* 339. 68–95.

[16] Gerlinger, P. 2012. Multi-Dimensional Limiting for High-Order Schemes Including Turbulence and Combustion. *Journal of Computational Physics*, 231, 2199–2228.

[17] Jack, S., C. Génin. 2014. Numerical and Experimental Investigation of Flow Separation in Ovalized Nozzles. In *4th Space Propulsion Conference.*

[18] Jack, S., C. Génin. 2015. Flow Separation Study in Stiff Ovalized Rocket Nozzles, Part II: Numerical Approach. In *51st AIAA Joint Propulsion Conference*. 2015-4154

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

[20] Lempke, M., R. Keller, P. Gerlinger. 2015. Influence of Spatial Discretization and Unsteadiness on the Simulation of Rocket Combustors. *International Journal for Numerical Methods in Fluids*. 79(9). 437–455.

[21] Marshall, W., S. Pal, R. Woodward, R. Santoro. 2005. Benchmark wall heat flux data for a go2/gh2 single element combustor. In *41st AIAA Joint Propulsion Conference*. 2005-3572.

[22] Oefelein, J. 2005. Thermophysical characteristics of shear-coaxial LOX–H 2 flames at supercritical pressure. *Proceedings of the Combustion Institute*. 30(2). 2929–2937.

[23] Pizzarelli, M. 2016. A CFD-derived correlation for methane heat transfer deterioration. *Numerical Heat Transfer, Part A: Applications.* 69(3). 242–264.

[24] Rudolph, M. 2016. Enhancements of the flow solver TAU for internal flows at supercritical pressures. In: *3rd International Conference of Fluid Flow, Heat and Mass Transfer (FFHMT'16).* Ottawa. Canada.

[25] Schneider, D., C. Génin. 2016. Numerical Investigation of Flow Transition Behavior in Cold Flow Dual-Bell Rocket Nozzles. *Journal of Propulsion and Power*, 32(5). 1212–1219.

[26] Seidl, M., M. Aigner, R. Keller, P. Gerlinger. 2017. CFD Simulations of Turbulent Nonreacting and Reacting Flows for Rocket Engine Applications. *The Journal of Supercritical Fluid.* 121. 63–67.

[27] Simsont, Y.H., P. Gerlinger. 2016 High Order Numerical Simulation of the Thermal Load on a Lobed Strut Injector for Scramjet Application. *International Journal of Numerical Methods in Fluids*. 82(7). 417–436.

[28] Suslov, D., J. Hardi, B. Knapp, M. Oschwald. 2016. Optical investigation of the LOx-jet disintefraction process at high pressure conditions in a LOx/H2 single coaxial injector combustion chamber . In *5th Space Propulsion Conference*.

[29] Suslov, D., J. Hardi, B. Knapp, M. Oschwald. 2015. Hot-fire testing of LOX/H2 single coaxial injector at high pressure conditions with optical diagnostics. In *6th European Conference for Aeronautics and Space Sciences (EUCASS).*

[30] Wagner, B. 2015. Rocket Propulsion in TAU, Ein Überblick über das Projekt ProTAU. In *64. Deutscher Luft- und Raumfahrtkongress.*

[31] Wang, Y.-Z., Y.-X. Hua, H. Meng. 2010. Numerical Studies of Supercritical Turbulent Convective Heat Transfer of Cryogenic-Propellant Methane. *Journal of Thermophysics and Heat Transfer.* 24(3). 490–500.

[32] Yoon S. H., C. Kim, K. Kim. 2012. Multi-dimensional Limiting Process for Three-dimensional Flow Analysis *Journal of Computational Physics*. 227. 6001–6043