3rd European Conference for Aerospace Sciences
Versailles, France, 6-9 July 2009

Improved Heat Transfer Prediction Engineering Capabilities for Rocket Thrust Chamber Layout

Maeding, C., Wiedmann, D., Quering, K., Knab, O.
Astrium GmbH Space Transportation, Launcher Propulsion, System Analysis,
81663 Munich, Germany

Abstract

The demand of a more comprehensive engineering tool for design and parametric investigations of the thrust chamber relevant heat transfer is pushing the improvement of coolant and hot gas side prediction tools. RCFS (Regenerative Coolant Flow Simulation) [Ref. 1], Astrium's in-house developed 1D tool to compute hot gas and coolant side heat transfer in a coupled approach, bases on the hot gas side Cinjarew approach, which has its origin in the late 1960. This tool was used as starting basis for the development and validation of a further improved method. Over the past years Astrium-ST has continuously expanded his knowledge in this field. In addition subscale hot firings, using different propellant combinations and injection conditions, relevant to open and closed cycle applications, were used for a the second RCFS generation – the RCFS-II

1. Introduction

The present propulsion development is more and more marked by a transition from a pure empirical - experimental work to a reasonable share between experimental work and computational design investigation. This approach requires adequate design tools covering all phases of engine development. The modern thrust chamber design work, including the thrust chamber sizing, hydraulic, heat transfer and combustion analysis, at Astrium-ST is based on a combination of commercial and in-house tools. The commercial CFX, the in-house Rocflam-II (Rocket Flow Analysis Module) and the RCFS-II (Regenerative Coolant Flow Simulation) programs form the main three pillars for a successful thrust chamber design in terms of combustion and heat transfer [Ref. 2].

Special interfaces between the different tools allow performing integrated sensitivity analyses using different tools for hot gas and coolant side simulations. An outstanding advantage is the direct transferability of the cooling channel geometry, established by RCFS-II layout calculations, towards the 2D Rocflam-II combustion code simulation as well as the 3D CFX coolant side heat transfer computation - presenting a high end heat transfer, combustion and hydraulic analysis approach. The possible coupling combinations of the three different tools are presented in the Figure 1.

The Rocflam-II presents a 2D axis-symmetric, finite volume Navier-Stokes code with an implemented chemical reaction model and parallel particle tracking. The reaction mechanism of bi-propellant systems can be approximated by a certain numbers of species. The tool allows to simulate multiple discrete phases and to compute the trajectories of these discrete phase entities by integrating their force balance taking into account various topics.

The subject of the paper is to highlight the main new features of Astrium's improved heat transfer engineering tool RCFS-II as well as the general approach used to develop and to verify this new method. One of the major development and improvement objectives is the still ongoing comparison of the calculation results with hot run test data in order to determine the degree of prediction accuracy and to determine the further direction of tool improvement.

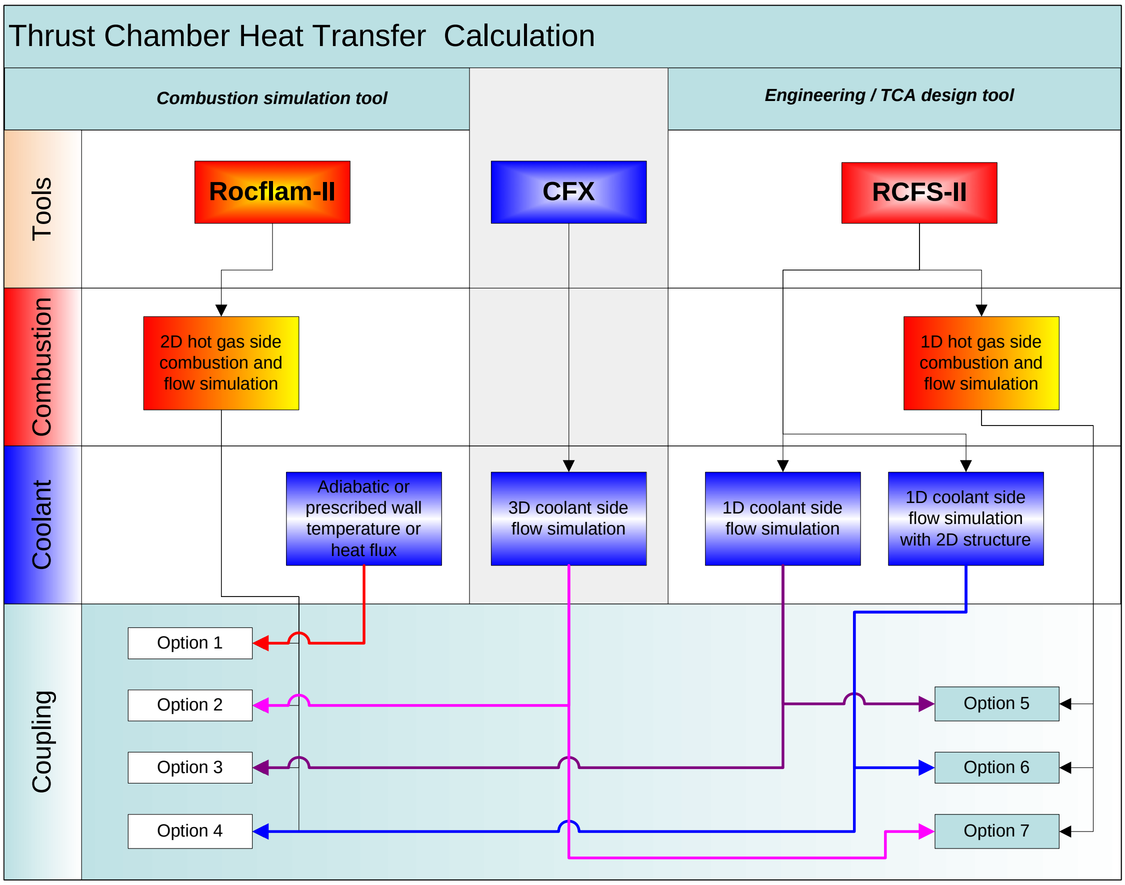

Figure 1: Rocket combustion and heat transfer simulation tools

2. General Heat Transfer Approach

In the early design and development phase of a new thrust chamber, an efficient simulation tool is needed to cover the main features of the new system without having test data available. The RCFS-II presents an upgraded version of the well known RCFS program [Ref. 1]. In order to fulfil the growing requirements, also with respect to new propellant combinations, a significant improvement was necessary. The RCFS program, used for many years, was a hot gas side Nusselt correlation based method (Cinjarew approach), reflecting all systems developed and used within Astrium-ST. However this 1D approach, which has its origin in the late 1960, requires a huge data base to reflect real system behaviour, due to the drawbacks of the pure Nu-correlation and different implemented empirical factors.
Over the past years Astrium-ST has continuously expanded his knowledge in this field. In addition subscale hot firings, using different propellant combinations (LOX-H2, LOX-CH4 and LOX-Kerosene) and injection conditions, relevant to open and closed cycle applications, were used. The employment of the initial RCFS code revealed a number of inadequacies, which were covered by applying experimentally based empirical correlations, in order to match realistic conditions. In order to fulfil these requirements and to increase the prediction precision already during an early thrust chamber development state, a higher sophisticated hot gas side heat transfer calculation method was implemented in the second RCFS generation – the RCFS-II. One major requirement was to increase the accuracy with respect to new propellant combinations, such as LOX-Kerosene or LOX-CH4 taking also into account injection relevant conditions.

In RCFS-II a so called non-interacting core flow - boundary layer (two stream model) approach is used (no iteration approach). The main parameter distribution, like static pressure, velocity, enthalpy and gas composition at the edge of the boundary layer is determined by a 1D flow approach throughout the whole thrust chamber, even if a 3 stream model with a 2D wall pressure distribution is used. That means at $y=\delta_{bl}$ the main parameters correspond to the 1D main flow. The 1D pressure distribution as well as main parameter definition is based on the Gordon McBride equilibrium combustion code [Ref. 4, 5]. Within the whole TCA and the boundary layer, equilibrium gas composition is used, since an adequate non-equilibrium code is currently not available. The whole picture of losses is covered by considering the combustion efficiency.

All relevant hot gas parameters at y=0 (at the hot gas wall) are calculated assuming also equilibrium combustion, applying $T=T_{wall}$ and a zero velocity. The parameter definition, including the chemical composition, at the hot gas wall is done using a T, p-problem definition file in the Gordon McBride combustion tool.

The consideration of a low temperature layer (film injection or element trimming) between the core and the boundary can be considered by implementing an additional stream between the core and the boundary layer. This requires an adequate mixing model between the core and the film layer, in order to generate the proper conditions for the edge of the boundary layer and relevant heat transfer, assuming a boundary layer mixture ratio equal to low temperature layer (see Figure 2). The mixing between the two flows is mainly driven by the matched on condition between core and film layer as well as the turbulence - defining the mixing length (effective film length).

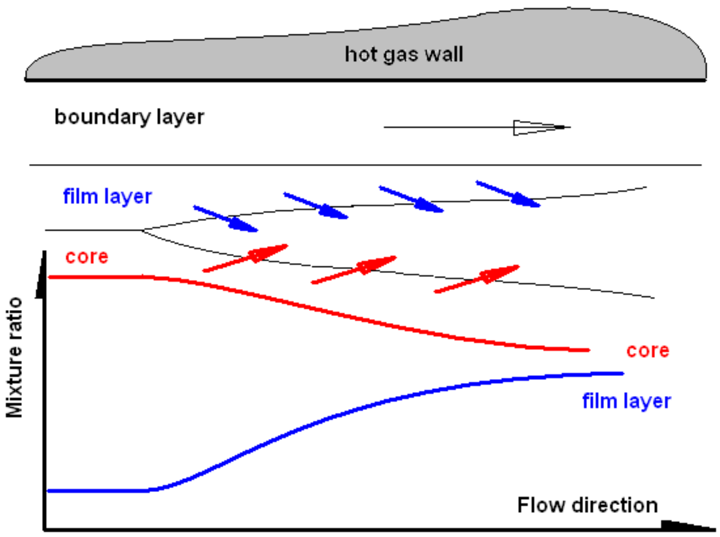

Figure 2: General flow picture if film cooling is applied

By implementing a boundary layer approach for heat transfer calculation a very important characteristic, which is missing using a simple Nu-correlation, appears. An increased cylindrical running length leads to a certain heat flux decrease while propagating downstream the cylinder towards the throat section. The Nu-based heat flux is marked by a constant heat flux along the cylindrical chamber section, neglecting the changed boundary layer along the chamber cylinder.

A major issue, while considering only the molecular motion, typical for an boundary layer approach, is to take into account the additional heat release, due to recombination processes in the cold near wall layer. Therefore the whole approach is based on a dissociation free system – intensifying the heat flux by a higher hot gas temperature keeping the core enthalpy unchanged.

The heat flux is determined by the following equation:

$$q = -\left(\frac{\lambda_m + \lambda_t}{c_p}\right) \cdot \frac{\partial J_\infty}{\partial y} = \alpha_{Turb} \cdot \rho_{av} \cdot v_\infty \cdot (J_\infty - J_{wall})$$

λ_m - molecular heat conductivity,
λ_t - turbulent heat conductivity,
c_p - average heat capacity,
α_{Turb} - dimensionless friction factor,
ρ_{av} - average density,
v_∞ - boundary layer edge velocity,
J_∞ - core flow enthalpy,
J_{wall} - hot gas wall enthalpy.

The dimensionless friction factor needed for heat transfer calculation depends on the temperature and velocity distribution across the boundary layer, assuming that the change of the velocity in this layer is less intensive than the temperature. Therefore the whole process is driven by a Pr-number <1 – typical for a compressible flow in rocket chambers.

The implemented model does not consider additional effects like exothermic monopropellant decomposition if MMH or UDMH film injection is applied. The impact of such additional effects is mainly driven by the reaction time with respect to the overall combustion process and mixing. Only if the monopropellant decomposition velocity exceeds the overall mixing and combustion processes a noticeable change of the heat transfer should occur. Tests with MMH showed, that at moderate temperatures of T=400K…700K monopropellant decomposition processes run very slowly and therefore can be neglected in relation to the chemical reaction of the bi-propellants [Ref. 3].

The current RCFS-II configuration does also not consider the droplet formation and vaporisation at injection. In the most cases the propellant injection takes place under supercritical conditions in terms of pressure or temperature leading to a zero surface tension. In this case the mixing of the injected propellants is characterized by a turbulence degree assuming a sudden combustion of the mixed propellants – infinite small combustion time. This viewpoint leads to a certain simplification keeping one for an engineering tool sufficient accuracy. Especially in rocket chambers with a low Mach number in the cylindrical section, where the main combustion take place, the gas propagation time is higher than the relaxation time to reach a new equilibrium state gas composition.

3. Hot Gas and Boundary Layer Conditions Implemented Modules

One major issue was to find a solution, which permits to consider the influence of injector and chamber dependent combustion efficiencies on the heat transfer. Already earlier investigations showed certain dependence between the completeness of the combustion process and the heat transfer. Due to the absence of a practicable method to consider the incomplete combustion in the actually local gas composition another path was chosen, taking also into account that the non-equilibrium combustion does not consider the overall picture of losses included in the combustion efficiency. The overall combustion efficiency presents a mixture of boundary, combustion, mixing and flow field losses without any possibility of clear differentiation between the different fractions. This unique solution allows taking into account the expected combustion efficiency level as essential heat transfer driver. So it became possible to perform sensitivity analyses in terms of combustion efficiency variation.

3.1 Injector

The total heat load in the cylindrical chamber section is mainly driven by the injector. The knowledge of accurate heat flux levels in that chamber region governs the tailoring of the cooling channels to optimize the pressure drop in the regenerative cooling circuit. Using the available in-house experience, the outer element row wall distance, the element type, the element size as well as the propellant state were identified as the major parameters. In order to solve this issue an additional method, based on a two layer mixing module induced by the outer row injection elements, was implemented eliminating also a major problem of the selected boundary layer method, which assumes a zero boundary layer thickness at injector face. Due to the zero boundary layer thickness at the injector head a very high heat flow is formed, which assumes that a uniform mixture ratio distribution is already present. In reality the heat flux at the injector head is mainly determined by the outer row injection element induced mixing process. This mixing of the jets is represented by a dedicated module applying a pre-defined turbulence degree.

In the case of an open cycle system both propellants will be injected under unmixed conditions. The transition to a closed cycle application requires that a pre-combusted gas with a higher temperature and certain gas composition will be injected, affecting the heat flux slope at the injector.

The current RCFS-II injector module allows also reflecting the impact of a simple coaxial or swirling injection on the heat transfer slope close to the face plate.

Figure 3 shows the impact of the injection angle, by using different swirl injectors on the slope of the heat transfer. A wide hollow cone propellant injection leads to a more steep heat flux slope, due to the more radial propellant interaction and mixing.

Transition from a pure, cold bipropellant injection to a mixed cold-pre-combusted propellant injection, typical for staged combustion applications leads to a rapid heat flux increase close to the face plate, due to the higher average injection temperature, keeping the average injection angle unchanged. Figure 4 shows the impact of a staged combustion like bi-propellant injection using the same injection element type.

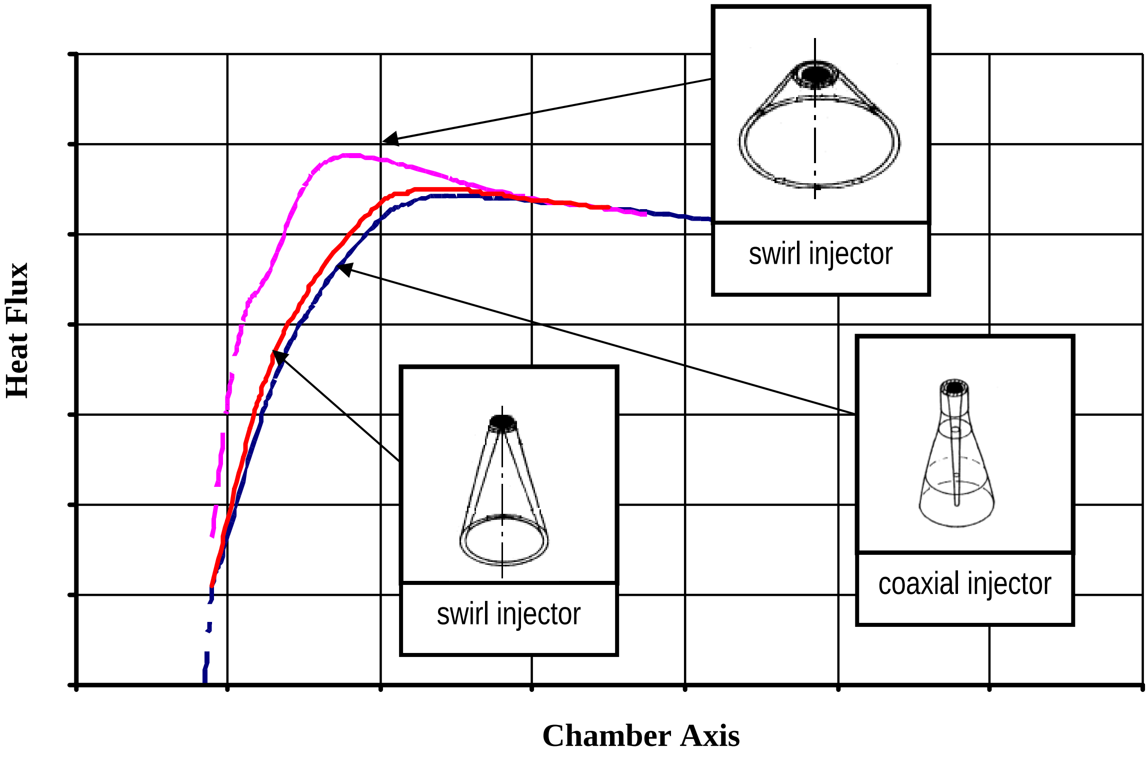

Figure 3: Injection angle impact on the heat transfer slope
(assuming equal combustion efficiency)

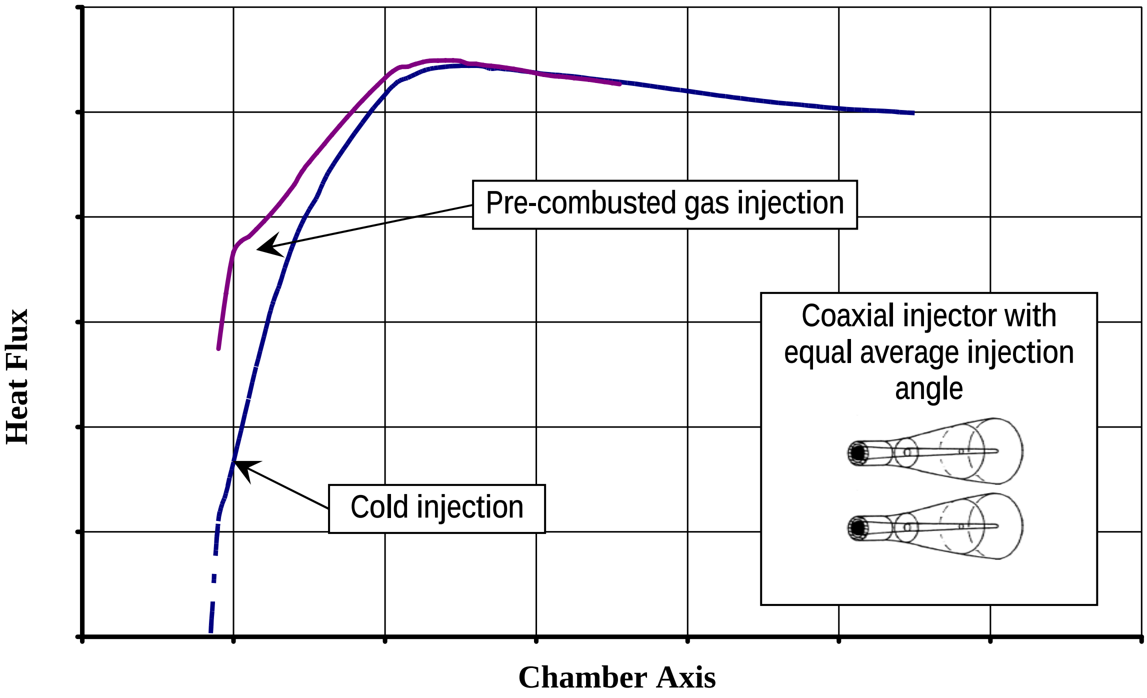

Figure 4: Comparison of cold and pre-combusted propellants on the heat flux evolution
(assuming equal combustion efficiency)

3.2 2D-nozzle expansion

Furthermore, different analytical analyses of the new RCFS-II tool revealed also the shortcoming of a 1D boundary layer approach in the supersonic nozzle section. Especially the strong wall pressure change of a parabolic nozzle, consisting of an arc and a parabola, leads to a strong change of the heat flux. Therefore a 2D wall pressure distribution approach was implemented providing a more realistic hot gas side heat transfer distribution.

A 2D wall pressure distribution, based on TDK [Ref. 6] or Rocflam-II calculations, can be used in order to reflect the specific near wall flow. The corresponding hot gas properties will be calculated using a modified Gordon McBride multi stream approach dividing the flow into three main streams (schematically shown in the Figure 5):

- 1D core flow,
- edge flow of the potential flow field,
- boundary layer flow.

The consideration of the 2D wall pressure takes into account the changed flow behaviour marked by a higher velocity in the near wall layer just after the throat in comparison to a 1D expansion. Close to the nozzle exit a lower near wall layer velocity with respect to the 1D expansion will occur. That means the gas composition in the near wall layer is not determined by the local nozzle cross section as used for a 1D expansion.

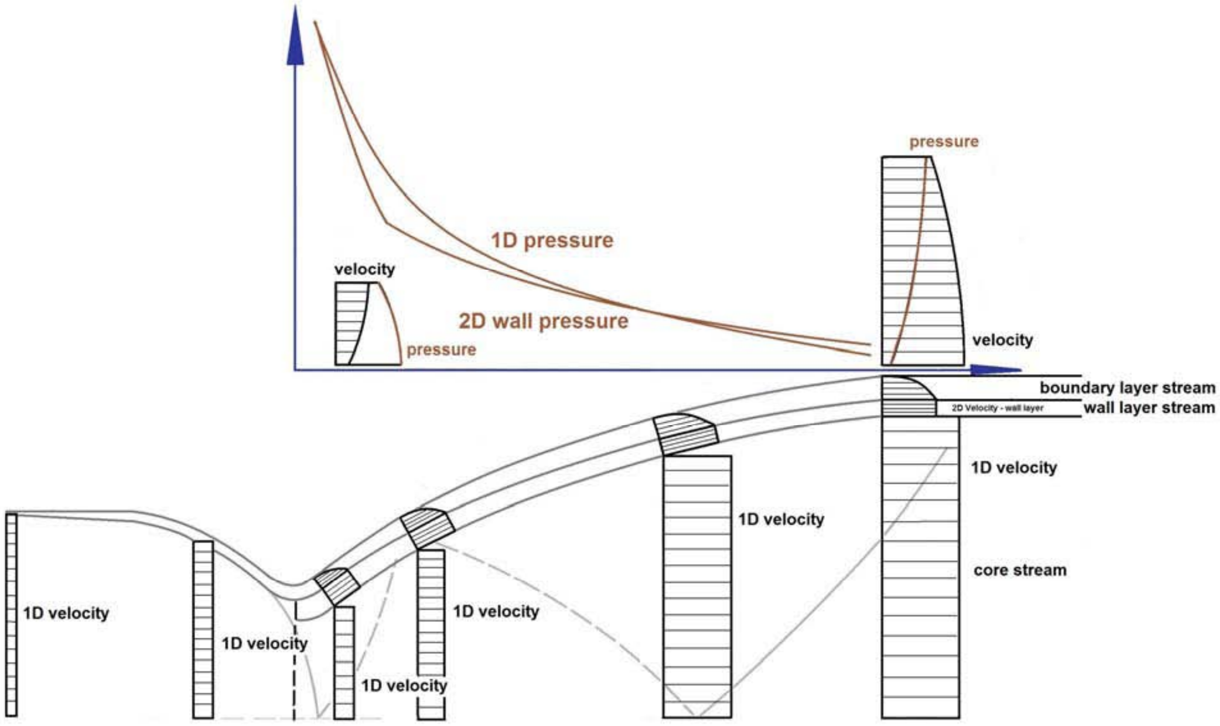

Figure 5: 2 and 3 stream model to reflect the 2D supersonic flow effect on the heat flux
(wall layer stream – TDK based pressure data)

The implementation of the above mentioned modules atop of the selected boundary layer approach results in a significant change of the whole heat flux picture, which gets closer to the reality one. Considering the whole thrust chamber (as presented in Figure 6) IV main regions can be identified. Area I – is mainly characterized by the injection, eliminating as far as possible the impact of the boundary layer. The area II is only marked by the boundary layer, taking into account also the combustion efficiency. The throat section area III is mainly marked by the combustion efficiency. The shape at this region is driven by the boundary layer and the cross sectional change of the chamber.

The supersonic nozzle section IV is marked by a rapid pressure change inducing a steep heat flux decrease towards the exit. A 2D approach results in a steeper slope after the throat, followed by a weak bend – transition to a more flat heat flux decrease in comparison to a 1D expansion. Such behaviour is typical especially for a non-streamline nozzle contour – profiled using the Rao-method.

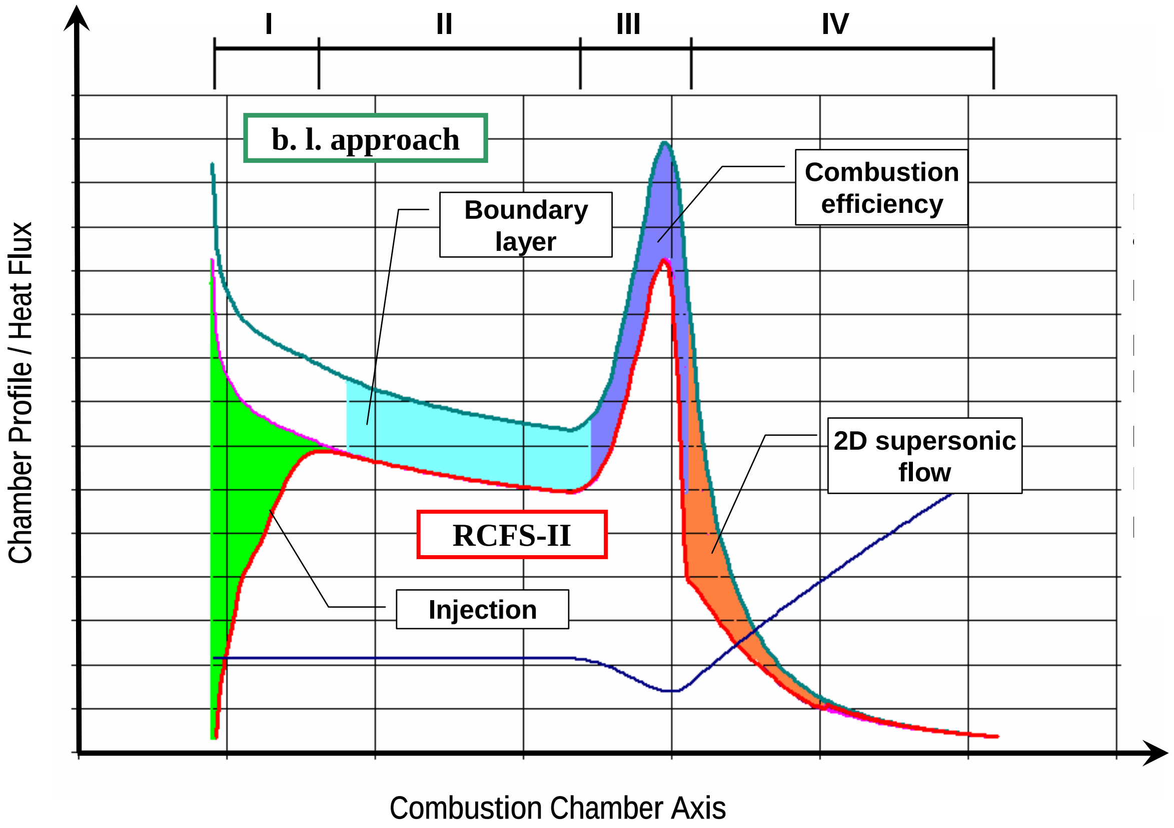

Figure 6: Visualization of implemented RCFS-II heat transfer modules compared to a pure boundary layer computation

4. Method Verification

One major objective in developing a new tool is the simultaneous verification and validation using all available test data, in order to recover still existing shortcomings and to determine the confidence level of the new method. However, additional points were disclosed, in order to cover all necessary fields of applications in terms of heat transfer predictions.

With respect to the recent main focus of development, the cross check activities were mainly dedicated to LOX-H2 and LOX-CH4, applying open and closed cycle thrust chamber configurations.

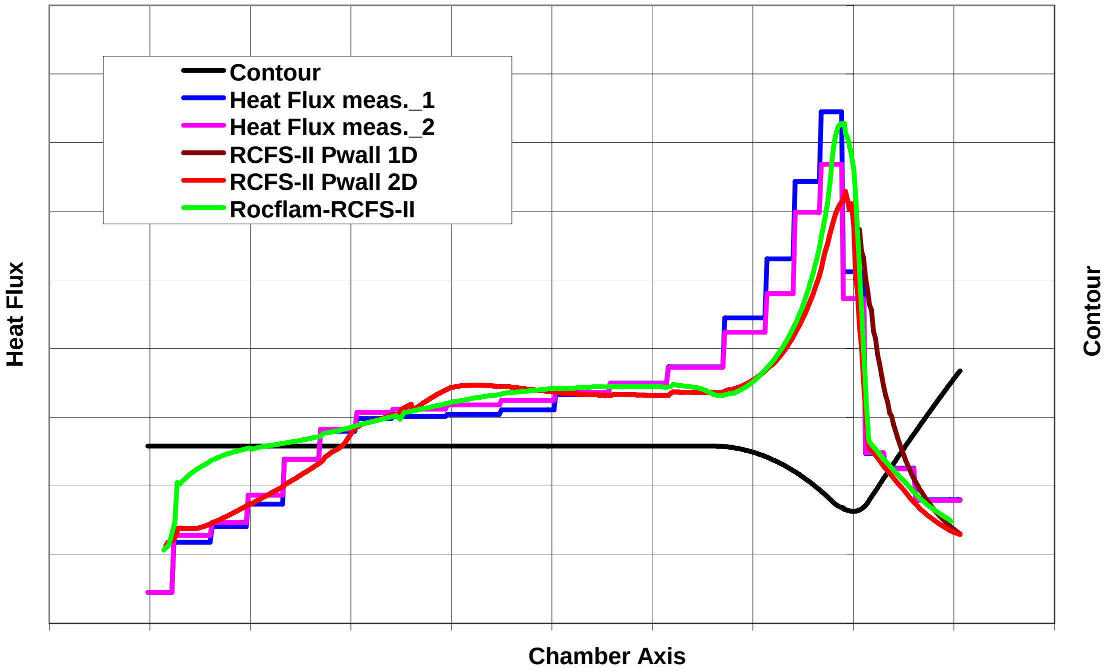

Figure 7: Heat transfer comparison using subscale test data

Figure 7 shows a comparison of the measured and calculated heat transfer on a standard, segmented subscale configuration. The combination of the different modules, considering the injection, combustion efficiency and the boundary layer, results in a quite good reflection of the measured heat transfer level and distribution without additional adaptations.

Especially at the combustion chamber entrance an acceptable heat flux evolution is obtained using the new RCFS-II. The comparison with a Rocflam-II combustion simulation further confirms the selected approach. Such analyses in defining the confidence level of the new tool version are very important for future full-scale design activities.

The Figure 8 below shows the comparison of the heat transfer calculated using the Rocflam-II and RCFS-II codes for a LOX-LH2 full-scale thrust chamber. The analysis of the RCFS-II data shows a sufficient agreement between both methods, necessary for an early thrust chamber development phase.

A further emphasis of the tool validation is in particular the analysis of close cycle engine applications with view on future propulsion systems using LOX-H2 and/or LOX-CH4. Therefore the available test data generated within the recent FLPP P8 test campaigns [Ref. 7], applying closed cycle schematics, will be also used to cross check the confidence of the upgraded engineering heat transfer code.

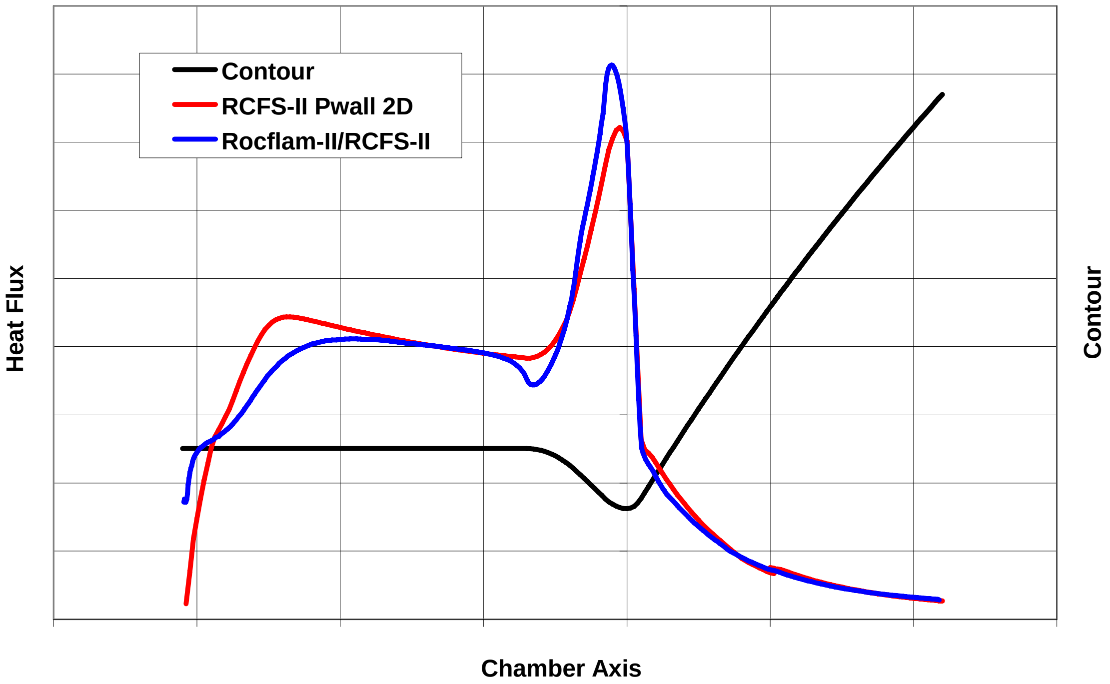

Figure 8: Heat transfer analysis on a full-scale TCA using Rocflam-II and RCFS-II

5. Conclusion

The improved RCFS-II boundary layer heat transfer engineering tool presents a powerful option covering a wide range of applications in terms of propellant combination and operational conditions.
The main advantage of the new approach consists of the renouncement of adaptation factors to reflect a more realistic heat transfer picture on the hot gas side. Especially the consideration of combustion efficiency on the overall heat flux distribution presents a major issue solved within the new tool modification.
The new RCFS-II allows considering also specific injection conditions, relevant for open and closed cycle injection.
Based on the still ongoing modification and validation, already a high accuracy in prediction of the heat flux evolution and level was demonstrated, without using any empirical data for adaptation – increasing at the same time also the user friendliness.
Nevertheless, further improvement is required in order to reflect all heat transfer relevant parameters. The current development is mainly focused on in-house thrust chamber configurations and injection systems. The film layer modelling presents beside the amelioration of the injector model one major field of RCFS-II improvement. This allows enlarging the field of application down to small thrusters marked by the presence of a film cooling, due to missing regenerative cooling.
The future development of the RCFS second generation is not only addressed to improve the hot gas side heat flux prediction accuracy. A further emphasis is laid on the improvement of the coolant side heat transfer modelling and prediction with respect to the different cooling channel configurations, taking also into account the impact of thermal stratification, curvature and coolant circuit inlet and outlet effects.

6. Reference

[1] A. Fröhlich, M. Popp, G. Schmidt, D. Thelemann, „Heat Transfer Characteristics of H2/O2 – Combustion Chambers“, AIAA 93-1826, 29th Joint Propulsion Conference, Monterey, Ca June 28-30/1993,

[2] O. Knab, M. Frey, C. Maeding, K. Quering, D. Wiedmann, W. Zeiss, "Progress in Combustion and Heat Transfer Modelling in Rocket Thrust Chamber Applied Engineering", AIAA-2009-5477, 2009,

[3] S. Boudigues, "Caracteristiques physiques et thermodynamiques de N2O4 et de MMH", Rapport technique n°36/6112 EH, ONERA, 23 juin 1987

[4] B. McBride, S. Gordon and M. Reno: "Coefficients for Calculating Thermodynamic and Transport Properties of Individual Species", NASA Technical Memorandum 4513, 1993

[5] S. Gordon and B. McBride: "Computer Program for Calculation of Complex Chemical Equilibrium Compositions and Applications", NASA Reference Publication 1311, 1994

[6] S.S. Dunn, D.E. Coats, G.R. Nickerson, D.R. Berker, " Two-Dimensional Kinetics (TDK91 PRO/X) Nozzle Performance Computer Programm", volume I,II,III, software and engineering Associates Inc., Carson City, Nevada, 21 November 1991

[7] R. Strunz et al, " Status of the LOX/GH2 Combustion Activities within the Furture Launcher Preporatory Programme FLPP", Space Propulsion Conference 2008, Heraklion, Greece.