Conjugate heat transfer applied to transitory analysis for rocket engine cooling systems design

Vincenzo Barbato[1,a*], Matteo Fiore[1,b], Francesco Nasuti[1,c]

[1]University La Sapienza of Rome, Department of Mechanical and Aerospace Engineering, Via Eudossiana 18, Rome, Italy

[a]barbato.1739632@studenti.uniroma1.it, [b]matteo.fiore@uniroma1.it, [c]francesco.nasuti@uniroma1.it

Keywords: Conjugate Heat Transfer, Liquid Rocket Engine, Regenerative Cooling Systems

Abstract. This study investigates the use of an in-house Conjugate Heat Transfer (CHT) numerical solver for the modelling of transient phenomena in liquid rocket engines active cooling systems. Heat transfer considerations place great limitations in the development of rocket engines and transient operative conditions are amongst the most critical. The current lack of models and numerical tools capable of accounting for the complexities of this time-dependent multi-physics problem, results in oversized cooling systems, long development times and increased risk of failure. The fine modelling of all the involved phenomena and their interaction with each other is crucial to achieve a correct prediction of the thermal fluxes and wall temperatures involved. Hence, CHT simulations are the state-of-the-art for this application. The CHT solver proposed in this work utilizes a partitioned coupling strategy where two extensively validated single-physics solvers exchange information through their interfaces at discrete time steps. A simplified version of the RL-10A-3-3A regenerative cooling jacket is considered as reference to test the strengths and the limits of this approach. Both a complete chilldown of the engine and part of the start-up transient have been simulated. The analyses performed show the ability of the solver proposed to deal with transient phenomena where fluid-structure interaction occurs. In addition, they provide a complete overview of the numerical issues related to the partitioned coupling approach. These preliminary results pave the way for further developments aimed at increasing the reliability of the solutions and extending the application field of the software developed.

Introduction

Transient heat transfer conditions can be encountered during thrust build-up (engine start-up) and engine shutdown in all rocket propulsion systems. These phases of the rocket engine operational life are amongst the most critical and thus, must be carefully analysed to avoid failures: a cooling system sized on the engine nominal conditions is not sufficient to guarantee overall safety and integrity. During the start-up or shut-down transients, the chamber or nozzle walls may reach temperatures higher than those recorded at steady-state, as well as experience large temperature gradients. Furthermore, transient analysis is particularly important for reusable engines, where the knowledge of the thermal loads involved, can be crucial to guarantee safety through the engine thermal cycles. The aim of the present work is to present a numerical solver for the CHT problem completely developed by the authors, to study transient phenomena occurring in rocket engine regenerative cooling jackets. This multi-physics solver is based on a partitioned coupling approach, where two separate single-physics solvers exchange information through the boundary conditions at their interfaces. Two transient conditions have been analysed, engine chilldown and engine start-up, assuming a simplified version of the cooling jacket of the RL-10A liquid rocket engine as a reference for both geometry and operative conditions.

Conjugate Heat Transfer

All active cooling techniques, such as regenerative cooling, are characterized by the interaction between the fluid coolant and the solid chamber walls. To model such interaction and to predict accurate values of temperature and heat flux, one must focus on conjugating the boundary conditions at the fluid-solid interface through coupled heat transfer analysis. Such a coupled field of study is termed Conjugate Heat Transfer (CHT) analysis [1]. Over the years, CHT has evolved as the most effective method of heat transfer study. A review of the coupling techniques currently adopted in the CHT community together with a thorough analysis of the stability of each method, can be found in the work of Verstraete and Scholl [2]. Amongst the various alternatives, a *partitioned approach with a serial coupling and a Dirichlet-Neumann boundary exchange method* has been selected for the present work. As suggested by M.B. Giles [3], this coupling strategy guarantees overall stability through the coupling iterations, thus resulting in the best choice for the first implementation of a new CHT solver.

Reference Case

The CHT approach for transient analyses developed in the present work has been tested on a simplified version of the cooling jacket of the RL-10A LOX/LH_2 liquid rocket engine [4]. Two transient operative conditions have been simulated: the complete chilldown of the engine and the start-up transient. The RL-10A has been chosen as reference because a great number of details are available for this engine. However, since the scope of the present work is to investigate the limits of the CHT approach proposed, a simplified version of the RL-10A cooling jacket has been considered to neglect multiple phenomenology arising from a complex geometry (i.e., curvature effects, variable cross section, etc..). As a result, a rectified version of the cooling jacket is considered, assuming a constant rectangular cross section for each of the 180 tubes that make up the cooling jacket, made of Type-347 Stainless Steel. Channel dimensions have been chosen equal to the dimensions of the real channel section at the nozzle throat [5].

Numerical Setup

A conjugate heat transfer model based on the coupled numerical integration of the Navier-Stokes equations for the coolant flow and the Fourier's law of conduction for the heat transfer within the solid has been adopted [6], exploiting in-house validated solvers [7]. The flow solver integrates the Reynolds Averaged Navier Stokes Equations, written in the conservation form, by a Godunov-type finite volume scheme, which is second order accurate in space. Turbulence is computed with the one-equation model of Spalart-Allmaras [8].

The fundamental hypothesis on which the coupling strategy is based, is that the heat capacity of the solid is much greater than that of the fluid. This translates in the assumption that when the flow field reaches equilibrium, the structural thermal conduction has not yet begun. In terms of the coupling logic, this implies that at each time step, the steady flow analysis and the unsteady thermal analysis are implemented in turn, as shown in Fig. 1. Steady flow field computed by an isothermal boundary condition and uniform temperature field in the solid are specified as initial conditions for the coupled flow-thermal analysis. Thereafter, the thermal boundary condition for the flow analysis is updated after every thermal conduction calculation for the solid. A maximum temperature variation ΔT is considered as stopping criteria for the transient solid simulation: when at least one cell in the solid domain changes its temperature by an amount greater than the ΔT selected, the unsteady thermal analysis is stopped and the solution is evaluated. From the analysis carried out, it results that this method is first order accurate in time. This is probably a consequence of keeping the heat flux constant during the unsteady thermal analysis in the solid.

Given the symmetry of the problem, to reduce the computational cost of the simulations, only half of the channel has been simulated. The solid is divided in five 3D blocks whereas for the fluid, a single 3D component is sufficient to characterize the entire domain. The number of cells adopted for the fluid domain and its cell-clustering parameters have been chosen to guarantee that $y_+ \leq 1$. Instead, for the solid domain, the discretization has been chosen to include a sufficiently high number of cells to reduce the mapping error in the coupling phase caused by non-matching grids.

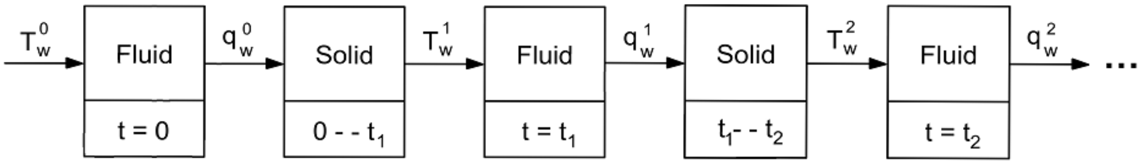


Figure 1 - Schematic representation of the coupling procedure adopted.

Results

To investigate the capabilities of the proposed CHT approach, a complete chilldown of the engine cooling jacket has been simulated. The analysis has been conducted assuming the following constant operative conditions for the fluid: inlet temperature T_{in} = 25 K, inlet mass flow rate m_{in} = 0.01 kg/s and outlet pressure p_{out} = 40 bar. Moreover, all external walls of the solid domain have been assumed adiabatic and the initial wall temperature is set to T_w = 240 K. With this study, the sensitivity of the solution to the variation of the stopping criteria (i.e., ΔT) has been investigated. Four different simulations have been realized with the same initial and boundary conditions, but with different simulation parameters. The results obtained showed that, for a simple problem as chilldown where the heat transfer process is monotonous, the variation of ΔT only slightly affects the overall time needed to simulate the same phenomenon. One of the major non-physical phenomena introduced by the discretization of the problem is the possible occurrence of *heat flux inversion*. In the case under analysis, heat flux inversion may occur towards the end of the chilldown process, when the temperature difference between the solid and the fluid approaches zero. When this happens, in the same time in which any cell of the solid changes its temperature of the ΔT required by the stopping criteria, the temperature of one or more of the wall cells, may become lower than the local fluid temperature. Thus, in the successive iteration, the heat flux locally changes its sign and rather than having the fluid cooling the solid, the inverse happens. Once heat flux inversion is triggered, solutions lose their validity and start oscillating because the heat flux reverses its sign alternatively from one coupling iteration to the other (Fig. 2). Furthermore, the use of decreasing values of ΔT suggested the possibility of a convergence

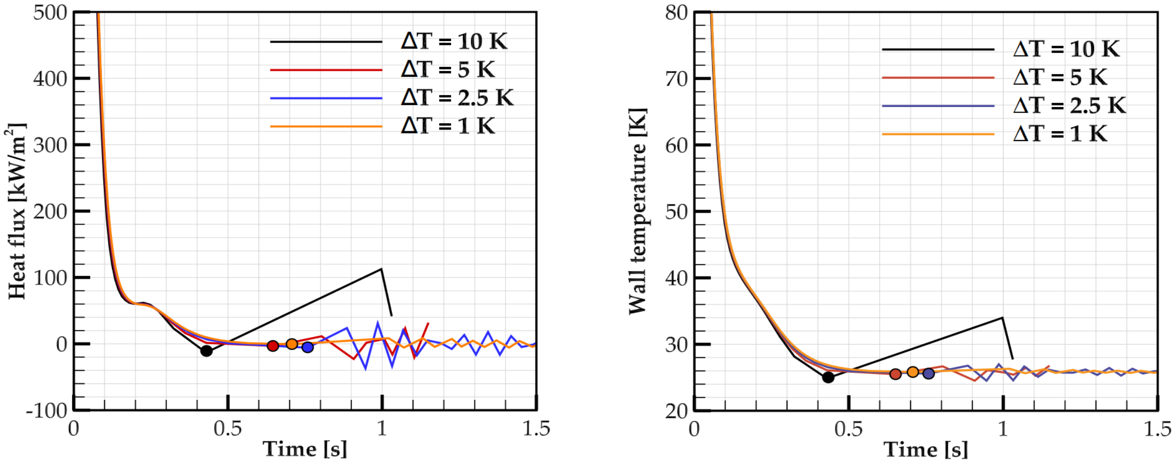


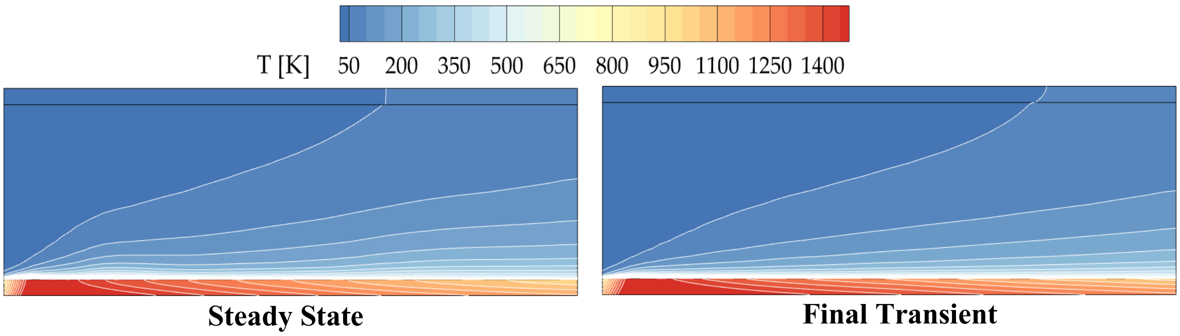
Figure 2 - Oscillations induced by the heat flux inversion phenomenon.

analysis analogous to the ones usually carried out for the spatial convergence. It has been found that the pseudo-order of convergence in time of this coupling strategy is approximately one.

Transient operative conditions in liquid rocket engines are generally much more complex than those characterizing a simple engine chilldown. During start-up and shut-down, cooling channels are subject to time-varying inflow and outflow conditions, as well as asymmetrical heating caused by the burning of fuel and oxidizer in the combustion chamber. In this work, part of the start-up transient of the RL-10A has been simulated. Several assumptions have been made to adapt the real start-up transient of the engine to the simplified geometry utilized. Not all the relevant quantities could be found in literature [4,5], thus a best-guess approach has been adopted to account for the missing or partial data. In particular, the heat flux value enforced on the hot-gas-side wall at each time $q_w(t)$ has been approximated by scaling the steady-state heat flux $q_{w,ss}$, through a function of the chamber pressure (Eq. 1) utilizing the information available in literature [5].

$$q_w(t) = q_{w,ss} \left(p_c(t) / p_{c,ss}\right)^{0.8} \tag{1}$$

It is crucial to emphasize that the scope of the present work is not to represent the real conditions of the start-up transient of the RL-10A. Instead, the aim of this further transient analysis is to show the ability of the solver to correctly represent the physical phenomena occurring in a much more complex scenario. Because of the wide temperature range involved in this simulation, an intermediate value of ΔT = 5K has been considered as stopping criteria, to reduce the number of couplings performed and the solutions evaluated. To show the consistency of the results obtained, in Fig. 3, the final solution of the transient simulation has been compared to a steady-state solution obtained with a parallel validated steady-state CHT solver [9]. In particular, the temperature field is shown along the channel symmetry plane. The presence of an intense monotonically increasing external heat flux, leads to a major modelling problem: the external heat flux is always underestimated. Indeed, the temperature distribution shows the same pattern suggesting that the solver is heading in the right direction. However, by constantly underestimating the real heat flux, lower temperatures are obtained with respect to the actual steady-state solution. To reduce this modelling error, a smaller ΔT should be adopted. However, for a simulation demanding as the one presented, this choice implies a non-negligible increase in the computational cost. Thus, to obtain a better trade-off between solution accuracy and computational time, a logic that automatically modifies the ΔT depending on the current solution of the problem, could be introduced. Given the multiple problems related to the constancy of the heat flux, such logic should be based on the percentage change, from one iteration to the next, of the magnitude of the heat flux itself.



Steady State

Final Transient

Figure 1 – Temperature field comparison for steady-state and final transient solutions.

Conclusions

This study presented a thorough analysis of the main limits of the in-house Conjugate Heat Transfer numerical solver developed by the authors. Overall, the partitioned approach proved to be a viable way of developing a new multi-physics solver, allowing for reuse of already existing validated tools. The results obtained show the capability of this CHT solver of dealing with the modelling of transient phenomena in liquid rocket engines active cooling systems. Despite the reported results being only preliminary, the methodology here proposed is quite general and can be easily extended to a wide range of problems where fluid-solid interaction occurs.

References

[1] John B., Senthilkumar P. and Sadasivan S., “Applied and theoretical aspects of conjugate heat transfer analysis: A review,” Archives of Computational Methods in Engineering, Vol. 26, 2019, pp. 475–489. https://doi.org/10.1007/s11831-018-9252-9

[2] Verstraete, T., and Scholl, S., “Stability analysis of partitioned methods for predicting conjugate heat transfer,” International Journal of Heat and Mass Transfer, Vol. 101, 2016, pp. 852–869. https://doi.org/10.1016/j.ijheatmasstransfer.2016.05.041

[3] Giles, M. B., “Stability analysis of numerical interface conditions in fluid–structure thermal analysis,” International journal for numerical methods in fluids, Vol. 25, No. 4, 1997, pp. 421–436. https://doi.org/10.1002/(SICI)1097-0363(19970830)25:4%3C421::AID-FLD557%3E3.0.CO;2-J

[4] Binder, M., Tomsik, T., and Veres, J. P., “RL10A-3-3A rocket engine modeling project,” Tech. rep., 1997.

[5] Binder, M., “A transient model of the RL10A-3-3A rocket engine,” 31st Joint Propulsion Conference and Exhibit, 1995, p. 2968. https://doi.org/10.2514/6.1995-2968

[6] Pizzarelli, M., Nasuti, F., and Onofri, M., “Coupled wall heat conduction and coolant flow analysis for liquid rocket engines,” Journal of Propulsion and Power, Vol. 29, No. 1, 2013, pp. 34–41. https://doi.org/10.2514/1.B34533

[7] Pizzarelli, M., Urbano, A., and Nasuti, F., “Numerical analysis of deterioration in heat transfer to near-critical rocket propellants,” Numerical Heat Transfer, Part A: Applications, Vol. 57, No. 5, 2010, pp. 297–314. https://doi.org/10.1080/10407780903583016

[8] Spalart, P., and Allmaras, S., “A one-equation turbulence model for aerodynamic flows,” 30th aerospace sciences meeting and exhibit, 1992, p. 439. https://doi.org/10.2514/6.1992-439

[9] Nasuti, F., Torricelli, A., and Pirozzoli, S., “Conjugate heat transfer analysis of rectangular cooling channels using modeled and direct numerical simulation of turbulence,” International Journal of Heat and Mass Transfer, Vol. 181, 2021, p. 121849. https://doi.org/10.1016/j.ijheatmasstransfer.2021.121849