METHOD FOR TRANSIENT AND STEADY STATE PARTITIONED FLUID-STRUCTURE INTERACTION ANALYSIS IN CONJUGATE HEAT TRANSFER FOR AERO ENGINES APPLICATIONS

Ute Israel[1], *Francois Cottier*[1]

MTU Aero Engines AG, Dachauer Strasse 665, 80995 München, Germany
ute.israel@mtu.de, francois.cottier@mtu.de

Christopher Hartmann, Jens von Wolfersdorf

Institute of Aerospace Thermaodynamics, University of Stuttgart, Germany
christopher.hartmann@itlr.uni-stuttgart.de, jens.von-wolfersdorf@itlr.uni-stuttgart.de

ABSTRACT

A method for fluid structure coupling for conjugate heat transfer for special demands of aero engines heat transfer is presented. It is based on an implicit partitioned approach and applicable for steady state and transient simulations. The different time scales of slow responding solid and fast responding fluid during a typical flight mission are considered by embedding a sequence of steady state CFD analyses to the solid thermal mission in the coupling process. A coupling control implementation designed to guarantee for engineering standard processes, methods and tools and able to deal with rotational symmetric 2D and 3D thermal models is presented. The structure is therein represented by a finite element (FE) model. Complex flows are modelled by CFD in multiple cavities as part of the fluid system. Other fluid system components (secondary air system) are modelled by an advective system. The coupled system is composed out of the three individual fields of FE, CFD and advective system. The method is experimentally validated and engine applications of an inner air seal turbine cavity and a compressor rotor cavity are compared using coupled and conventional analyses.

KEYWORDS

fluid structure interaction, conjugate heat transfer, coupling, partitioned approach, multiphysics

NOMENCLATURE

CFD	computational fluid dynamics
CHT	conjugate heat tranfer
FSI	fluid structure interaction
j	number of subiteration
$\mathcal{F}$	fluid system
$\mathbf{n}$	normal vector
$\mathbf{q}$	heat flux
ref	reference
$\mathcal{S}$	structure system
t	time
$\mathbf{T}$	temperature
α	heat transfer coefficient
ε	convergence limit
$()_F$	corresponds to fluid domain
$()_S$	corresponds to structure domain
$()_\Gamma$	corresponds to interface Γ

---

[1] both authors contributed equally

INTRODUCTION

Component life and life assessments in aero engines are strongly dependent of material temperatures and, therefore, need reliable material temperatures prediction for typical flight missions. Conjugate heat transfer (CHT), the interaction between material and fluid temperatures has a significant impact on heat transfer in several regions of an aero engines. Flow conditions in cavities influence strongly the material temperatures. Thermal interaction appears e.g. for secondary flow cavities in compressors and turbines. The representation with conventional models using the advection system to simulate cavity and secondary air system flow is often limited, correlations between fluid and solid configured with manual input from CFD are tedious to model and matching activities are effort intensive. Coupled multiphysics simulations with direct and automated application of the convective boundary condition including three systems cavity fluid, structure and advection system can significantly improve quality and efficiency of conjugate heat transfer analyses. Research about fluid structure interaction and further multiphysics coupling is done since many years (Felippa, Park, and Farhat 2001). Also CHT and the stability of respective numerical approaches is deeply examined (Errera et al. 2019; Radenac, Gressier, and Millan 2014) and published. Application for aero engines can be found in Ganine et al. 2012; Schindler 2022; Schindler, Brack, and von Wolfersdorf 2019; Sun et al. 2011.

We present a coupling control method for partitioned implicit FSI analysis with references to its validation and show engine applications. The coupling framework is capable to handle commercial black box solvers as well as open software and is designed to be extensible to further solver and mapper tools. This guarantees for keeping approved industrial engineering standards and for specialized software and know how. With this process significant improvement in effort and quality was achieved due to the irrelevance of correlations, the automation of the process and the representation of for one-way and two-way coupling.

Due to special requirements of coupled CHT analysis for aero engine flight missions the approach differs from other metods coupling methods. A major special requirement discussed in this contribution is, that the underlying method must provide a adequate time stepping scheme in consideration of a good performance. A flight mission is several hours long and characterized by different operation points with different characteristics and underlies a high rate of thermal condition changes. Typically, the thermal mission analysis of solid material is resolved in a huge amount of time steps to optimally represent the transient behavior during a flight cycle. However, fluid temperatures show much faster reaction than material temperatures and time scales differ for the respective domains. To avoid coincident time discretization steady state CFD analyses for selected operation points are used in the presented method to provide boundary conditions for the solid. Further requirements are to include multiple cavities modelled through independent CFD domains as well as the advective fluid network and its interaction with cavity flow and solid into the coupled simulation. Methods for data transfer between rotational symmetric 2D solid models and 3D models with 3D CFD models are required.

The above industrical objectives wrt. the restrictions of correlations and matching activities are also a motivation of Sun et al. 2011. Also the issue of a transient cycle for 2D-3D coupling of aero engines industrial applications is treated using a modular partitioned coupling framework. Mechanical coupling (blade tip deflections) is included here. The incorporation of an advective system into the coupled system is not addressed.

NUMERICAL METHOD

The Partitioned Approach

For coupled problems coupling conditions on the solid fluid interface must be fulfilled. For CHT the continuity of temperatures $\mathbf{T}$ and heat fluxes $\mathbf{q}$ are required on the common interface. The interface Γ is the common boundary of solid and fluid domain with a normal component $\mathbf{n}$. The subscripts F and S denote the fluid and structure domain respectively.

$$\mathbf{T}_{S,\Gamma} = \mathbf{T}_{F,\Gamma} \tag{1}$$

$$\mathbf{n}_{S,\Gamma}\mathbf{q}_{S,\Gamma} = -\mathbf{n}_{F,\Gamma}\mathbf{q}_{F,\Gamma} \tag{2}$$

In the partitioned coupling approach the single physicals fields are independently modelled and solved. Coupling is represented by the exchange of boundary conditions. This enables the usage of approved software solvers. The structural field or structural solver operations can be formulated as nonlinear function $\mathcal{S}$ with a heat flux boundary condition as input and temperatures as output. In the same way the fluid field is denoted as function of temperatures with heat flux as output.

$$\mathbf{T}_{\Gamma} = \mathcal{S}(\mathbf{q}_{\Gamma}) \tag{3}$$

$$\mathbf{q}_{\Gamma} = \mathcal{F}(\mathbf{T}_{\Gamma}) \tag{4}$$

So the coupled problem can be represented as combination of this functions:

$$\mathbf{T}_{\Gamma} = \mathcal{S} \circ \mathcal{F}(\mathbf{T}_{\Gamma}) \tag{5}$$

Implicit methods aim convergence of this coupled problem in each in time step by solving the equation in a staggered algorithm. The method described in this paper is based on fix-point-iteration corresponding to the following formulation:

$$\mathbf{T}_{\Gamma}^{(j+1)}(t) = \mathcal{S} \circ \mathcal{F}(\mathbf{T}_{\Gamma}^{(j)}(t)) \tag{6}$$

Continuity of temperatures is reached when input and output temperatures do not deviate anymore. As convergence criteria we can define a infinity norm or root mean square norm on the difference of the temperature fields from succeeding iterations and state that convergence is achieved if a certain limit ε reached.

$$\Delta\mathbf{T}_{\Gamma}^{(j)} = \mathcal{S} \circ \mathcal{F}(\mathbf{T}_{\Gamma}^{(j)}) - \mathbf{T}_{\Gamma}^{(j+1)} \tag{7}$$

$$\|\Delta\mathbf{T}_{\Gamma}\|_{\infty} \leq \varepsilon \tag{8}$$

$$\|\Delta\mathbf{T}_{\Gamma}\|_{L2} \leq \varepsilon \tag{9}$$

The analogue equation and criteria as for temperatures apply for heat flux, convective boundary conditions and advective system boundary variables as mass flow, pressure and fluid temperatures.

Convective Heat Transfer Using Robin Boundary Conditions

In CHT the boundary conditions on the structure coming from fluid analysis can be imposed as heat flux or convective Robin boundary conditions given by a heat transfer coefficient α and a reference fluid tempaeraturе $\mathbf{T}_{ref}$. Coupling by Robin boundary conditions is desribed and examined in the collaboration work of Schindler 2022; Schindler, Brack, and von Wolfersdorf 2019. This has shown better stability with the current implementation.

Two possibilities exist to determine heat transfer coefficient and reference temperature. The first is to fix a user defined α and to calculate the corresponding $\mathbf{T}_{ref}$ from heat flux and wall temperature on the interface from the last iterations using the definition for the local heat transfer coefficient

$$\mathbf{q}_{\Gamma}^{(j)} = \alpha(\mathbf{T}_{\Gamma}^{(j)} - \mathbf{T}_{ref}^{(j+1)}) \tag{10}$$

However, better results, stability and convergence is observed for more physical values of the heat transfer coefficient. Flow is varying strongly along the interface and so near wall heat transfer is also location and time dependent. Considering this, the second approach is to set a near wall fluid temperature as a reference temperature and the calculation of the local heat transfer coefficient from the above equation. The law of wall according to Kader 1981 is used for extracting α and $\mathbf{T}_{ref}$ from CFD results.

Coupling Process And Time Stepping Scheme

A long transient thermal solid analysis for flight mission with time discretization adjusted to solid behavior cannot reasonably be coupled to a transient CFD with coincident time stepping due to performance reasons. Fluids react much faster and need therefore different time discretization. To overcome this problem several steady state or stationary CFD analyses of characteristic operation point are coupled to the thermal solid mission at the respective time steps. It is an efficient approach since much fewer steps need to be considered in CFD. Between CFD embedding time steps can lie several solid time steps. The embedded CFD analyses deliverer boundary conditions for the solid from which the boundary conditions for the solid time steps located in between them are derived. Due to the character of a typical flight mission which can be subdivided in different phases this approach is well suitable. When condition change rates are high and nonlinear CFD analyses are embedded more often whereas long phases with low condition changes need less CFD. The CFD embedding points are therefore irregularly spread over the mission. Te use of embeddded steady state CFD analyses is also described in Schindler 2022; Sun et al. 2011

In two-way interaction or strong coupling the fluid field has impact on the solid thermal behavior and solid temperatures vice versa influence the fluid field. On the other hand, in one-way interaction or weak coupling the impact of solid temperatures on the fluid field is insignificant for some engine typical applications. Two-way interaction can be represented iteratively staggered realizing subiterations. Since different amounts of solid time step lie between CFD embedding time steps a subiteration goes back several solid time steps. Convergence in such a time segment consisting out of a group of time steps and bounded by two CFD embedding time steps is aimed to guarantee for reliable transient mission temperatures. As corresponding to the implicit coupling approach progression in time is provoked after convergence is reached within the subiterations. This means a progression until the next CFD embedding time point is done passing the next time segment as depicted in figure 1. In the first subiteration loop a constant fluid boundary condition (also denoted as fluid load) is assumed for a time segment. For the subsequent subiterations a first guess of the fluid load for the next CFD embedding time step is known and these loads can be used to assess the fluid load for the time steps of the segment. This is done by linear load ramping. This time stepping scheme is also presented in Schindler 2022.

To improve the prediction of fluid load within a segment a transition curve concept has been worked out to enhance the method. Transition curves are normalized nonlinear functions of time representing the alternation of a variable between two defined time points. These curves result from engineering experience from analytics and measurements of aero engines and are typical input for transient thermal calculations. Using transition curves instead of linear load ramping enables the prediction and scaling of fluid load as depicted in figure 2. This concept has the potential to reduce the amount of CFD embedding time points since nonlinear behavior between to CFD embedding time steps can be approximated. Furthermore, a prediction for the first subiteration can accelerate convergence. Since CFD is the most costly component of the coupling process significant performance improvement is expected.

The time stepping method from figure 1 can also be used for steady state thermal analysis. Steady state solid models can coupled in the same way as transient ones. Either by using subiteration around one singe steady state solid time step and one single outer iteration for achievement of convergence or by defining a pseudo time and using outer iterations for convergence also with typical one steady state time step between two CFD embedding ones.

TECHNICAL IMPLEMENTATION

Single Field Models And Simulations

The coupling method is applied on industrial application using solid thermal model designed with either 2D rotational symmetry and cyclic boundary conditions or as 3D models. They are solved using the open source finite element solver Calculix. The cavity flow is always modeled as 3D domain and solved by the finite volume method using Ansys CFX. The secondary air system is modelled as advective fluid network also using Calculix.

Coupling Control Software

Coupling is realized by the in-house coupling control software CoSMiX. It provides central control methods for managing subprocesses as solver call, data transfer, convergence and time control. Convergence and time stepping settings can be configured by the user. The design consists out of this central control routines and independent modules for solvers which are also configurable. For extension to different solvers and physics new modules can independently be developed and easily be managed by the same central control routines. The coupling framework is capable to handle the engineering standard solvers, commercial black box solvers as well as open software and extensible to further tools. Extensions for mechanical coupling with defections and for full integration of secondary air system are planned.

For the above method the solid model prescribes the time. The solid can be declared as time master, so that coupling control adopts its time scheme and can manage the coupled and non-coupled boundary conditions for the CFD analysis of the different operation points.

Data transfer and mapping is also realized in independent modules. Mapping between non-matching grid interfaces of 3D models and from 3D to 2D models and back is implemented using an averaging method. Data tranfer is also implemented for the interface variables of

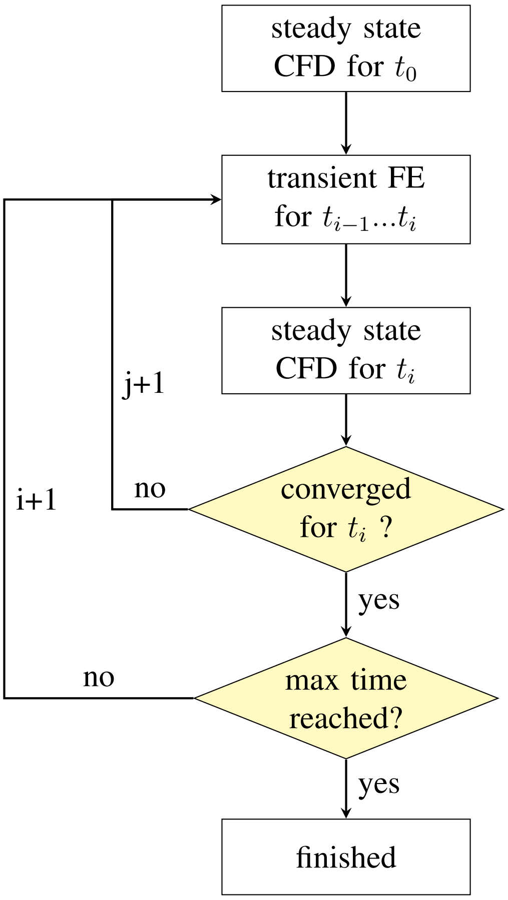


Figure 1: Implicit partitioned coupling process

the advective system mass flow, pressure and fluid temperature. Global convergence control is carried out using all these interface variable.

EXPERIMENTAL VALIDATION

The coupling method was validated with experimental data for conjugate heat transfer. A V-ripped channel subject to different boundary conditions has been transiently measured and simulated. The coupling method showed very good agreement with the measurements and with benchmark simulations using Ansys CFX CHT. For detailed description of validation it is referred to the collaboration work of Hartmann et al. 2023.

ENGINE APPLICATIONS

Inner Air Seal Turbine Cavity

A low pressure turbine inner air seal from a modern engine has been used to evaluate the coupling methods. Inner air seals flow structure is quite complex including many recirculation and interactions with main gas path as shown in figure 3, and is highly influenced by the real geometry. This type of flows has been typically simplified in standard thermal models using 1D flow network and correlation based heat transfer coefficients. The use of high fidelity 3D CFD methods enables to describe much more precisely the flows, and therefore the interaction

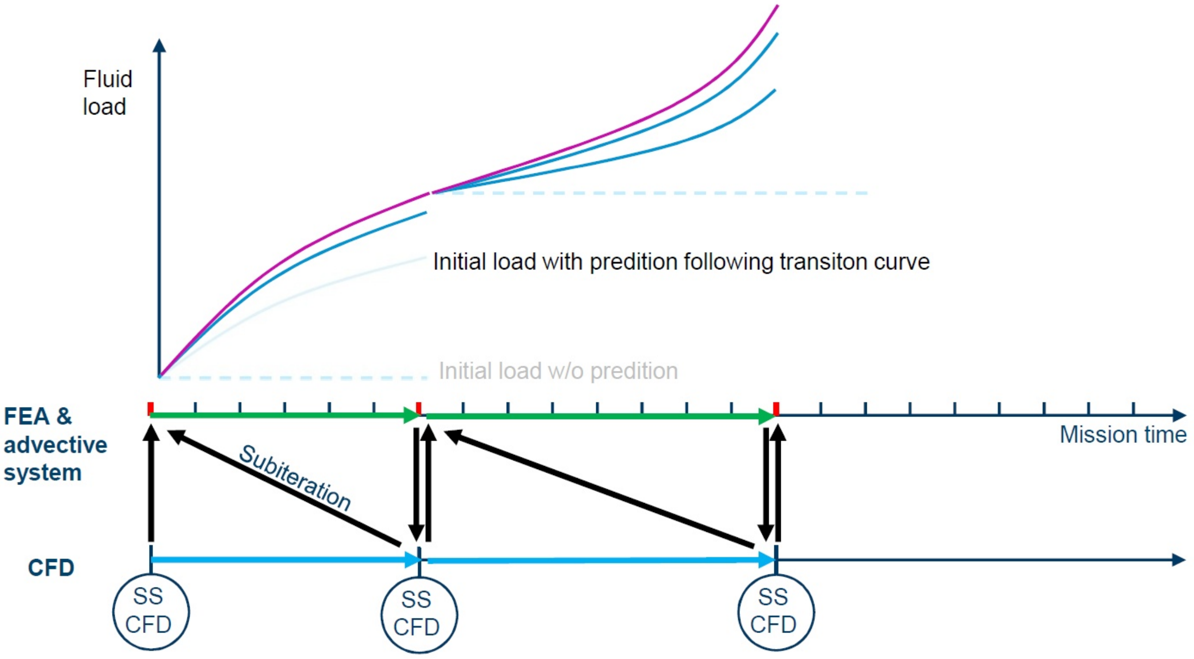
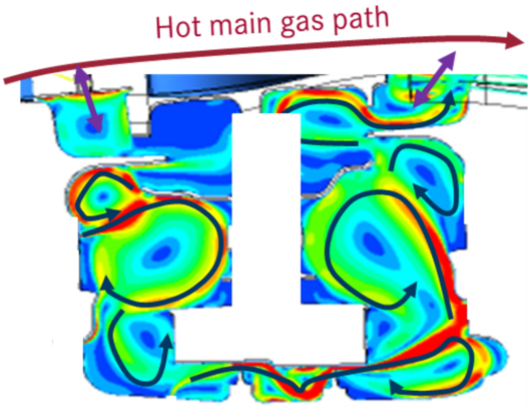


Figure 2: Fluid load application using transition curves

between secondary flows and main gas path, also by a better prediction of hot gas ingestion. A good prediction of the flow has become more crucial during the last years and the development of high speed low pressure turbine, where disc-blade attachments require particular attention.



$$V_{Ax,Rad} = \sqrt{V_{axial}^2 + V_{radial}^2}$$

Figure 3: Complex typical inner air seal flow structure showing meridional velocities

Measurements in the blade foot area, near dead and life rim, from engine test has therefore been compared to coupled numerical results, and shown in figure 4. The comparison shows an agreement below 3% which is a very good agreement for a numerical model that do not include matching factorization. This represents nearly a factor 2 in the uncertainty reduction of the numerical model before matching. In addition, by comparing with the measurement uncertainty that may be up to 0.2% and to the uncertainty related to circumferential variations that may reach 1% for the evaluated region, the agreement remains in a very acceptable range. This is a great improvement, however as explained before, the transient evaluation is of major importance for the structure mechanical assessments.

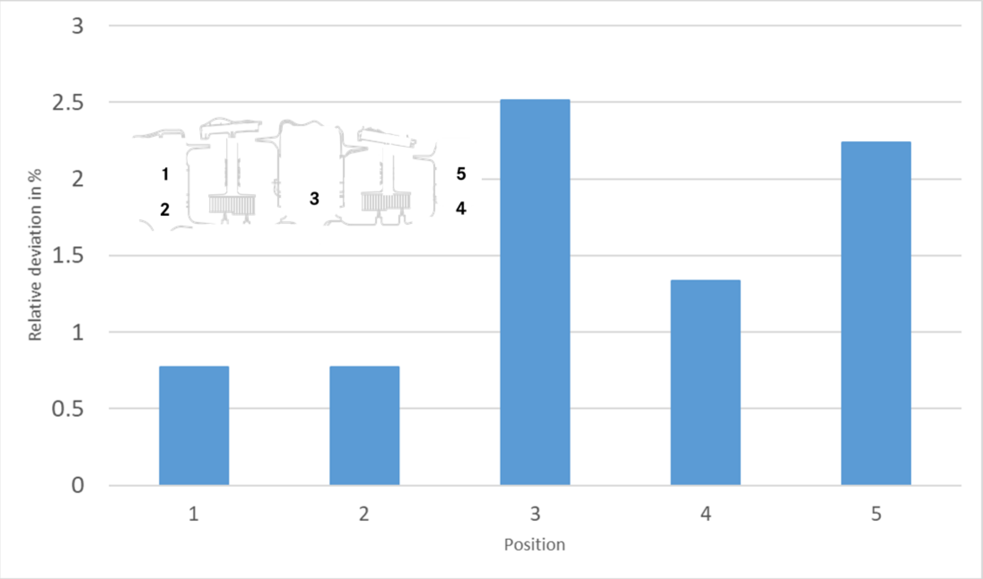


Figure 4: Relative deviation of material temperatures between measurements and calculations at steady state take-off

A typical snap-acceleration (from Idle to Take off) has been used to compare the numerical predictions to the measurements and are shown in figure 5. The position 3 has been transiently evaluated and compared to measurements. The comparison shows differences below 5% which is a very good agreement for a thermal model before any matching activities. The measurements have been used as reference for the normalisation of both measured and calculated temperatures.

It has been observed that the initial time period (prior to 10s) of the acceleration is well simulated. After 1000s simulated acceleration time, the takeoff temperatures are not fully stabilized and the difference are 1% higher than the steady state take off presented in figure 5. The transient phase is well reproduce, with a slightly too early temperature increase and a slightly lower rate in the temperature rise.

The calculation time has been kept acceptable for an industrial use with a total of 1728 CPU Hours for 1000 s real acceleration time.

These results give a very promising comparison between the coupling method and the measurements, and may clearly reduce matching activities that are known to be very time intensive.

Compressor Rotor Cavity

A second area in turbojet engine that is challenging to simulate for thermal model are closed rotating cavities, that are typically found in high pressure compressors. As explained in Owen and Rogers 1995, the flows in such cavities are chaotic, unsteady, and highly driven by the

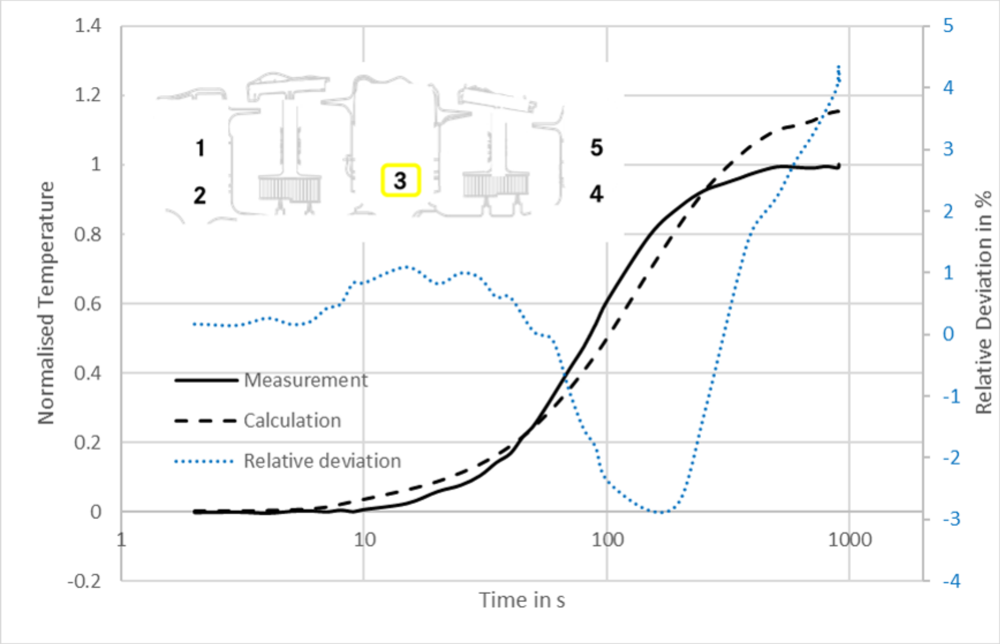


Figure 5: Normalised temperature and relative deviation between measurements and calculations at position 3 during snap-acceleration

temperature difference between the cold axial bore flow and the hot shrouds. Therefore, two-way interaction appears between the wall temperature and the flow structure. Such flows are thus predestined to be simulated with coupled system.

A two-cavities test rig has been measured. For this measurements, the uncertainty (relatively to the temperature range) has been evaluated to 2%. A monolithical CHT calculation has been carried out with Ansys CFX and show a very good agreement, with differences below 5%. This aims to generally show the high potential of CFD and coupled calculations to reproduce such flows and material temperatures as presented in figure 6. This validated monolithical method is taken as reference for applications without measurements availabilities. However, in realistic industrial situation, monolithical CHT calculations are not well adapted as standard thermal simulation are build up in complex FEM global models.

Another more realistic cavity system has been investigated and coupled simulations using CoSMiX are compared with monolithical CHT with Ansys CFX as no measurements are available for this configuration. The CoSMiX results are shown at different states of convergence progress, after the first iteration, after 15 and after 35 ‘loops’ between CFD and FEM.

The couple-back effect is clearly visible in figure 7 with the agreement improvement between the simulation after the first loop and after 35. After the first loop, a maximum disc temperature difference of 8.43% has been observed. This difference falls to 2.5% at loop 15 and 1.6% at loop 35. This enlightens the high impact of coupling the material temperature and the flow calculation. Standard thermal models that nowadays often use CFD inputs may not be able to take this phenomena into account, and therefore will require higher thermal matching effort, and higher discrepancies between design models and validated models.

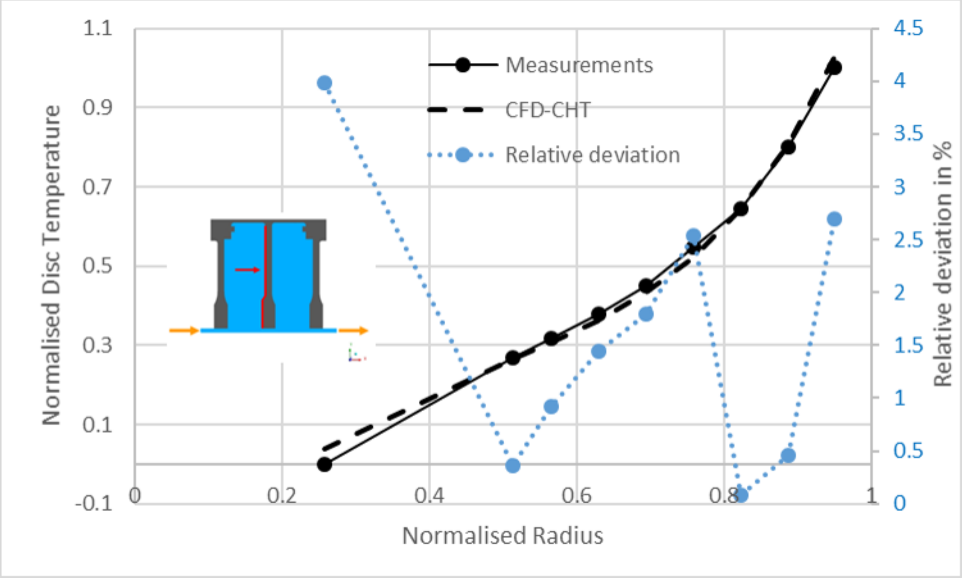


Figure 6: Normalised temperature comparison of CFD-CHT calculations with measurements

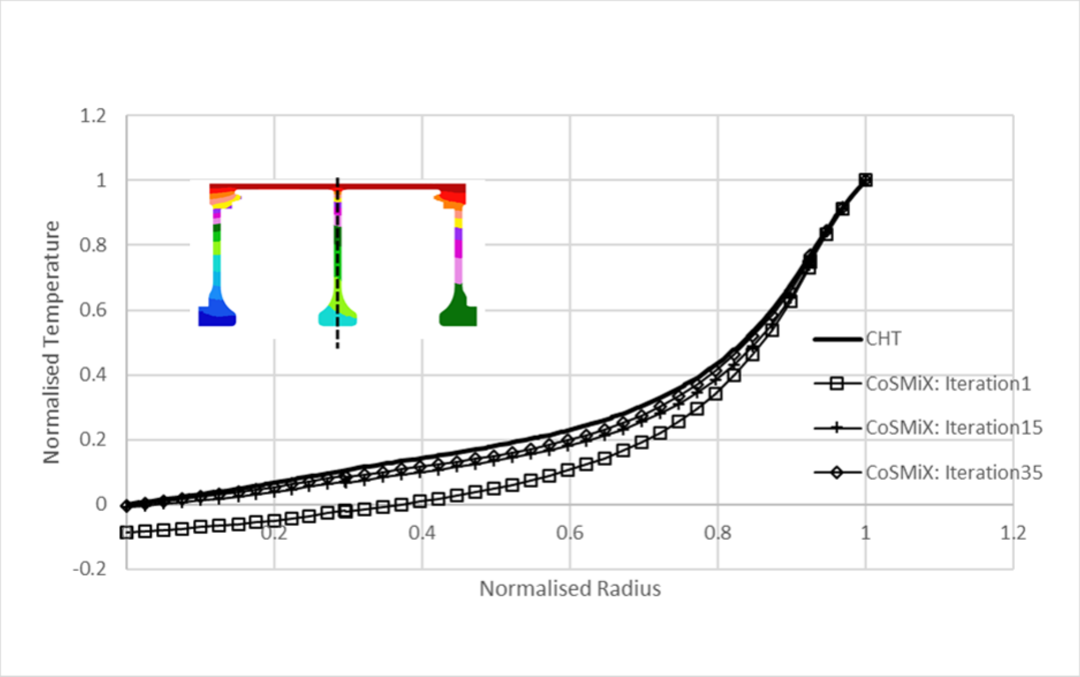


Figure 7: Normalised temperature comparison between CFD-CHT and CoSMiX

CONCLUSIONS

A partitioned implicit method for coupled simualtion of conjugate heat transfer for transient aero engine flight missions was presented. The method is based on embedding a number of steady state CFD analysis of selected operation points into a solid thermal analysis with a by far larger number of time steps. With the method the direct and automated boundary condition application of fluid load is enabled. The technical implementation in a coupling framework was explained. The validation of the presented method referenced in Hartmann et al. 2023 has shown good agreement.

Effort reduction was achieved with the full automated process implementation and transient and steady state engines applications have been shown. A comparison with convential model results after measurement calibration has prooven that physical mechanisms can be represented and showed the acchievement of our aspired objective of 50% reduction of deviation. Further performance improvement is is aimed in follow-up activities by introduction of transitions curves for engine examples and relaxation methods.

ACKNOWLEDGEMENTS

The project was partially participant of the joint research project KuepLe in the program AG Turbo funded by German Federal Ministry for Economic Affairs and Climate Action (BMWK).

REFERENCES

Errera, Marc-Paul et al. (2019). “A single stable scheme for steady conjugate heat transfer problems”. In: *Journal of Computational Physics* 394, pp. 491–502.

Felippa, Carlos A, Kwang-Chun Park, and Charbel Farhat (2001). “Partitioned analysis of coupled mechanical systems”. In: *Computer methods in applied mechanics and engineering* 190.24-25, pp. 3247–3270.

Ganine, Vlad et al. (2012). “Coupled fluid-structure transient thermal analysis of a gas tur-bine internal air system with multiple cavities”. In: *Journal of Engineering for Gas turbines and Power* 134.10.

Hartmann, C. et al. (2023). “Experimental validation of a numerical coupling environment between FEMand CFD”. In:

Kader, B.A. (1981). “Temperature and concentration profiles in fully turbulent boundary layers”. In: *International Journal of Heat and Mass Transfer* 24.9, pp. 1541–1544. ISSN: 0017-9310. DOI: `https://doi.org/10.1016/0017-9310(81)90220-9`. URL: `https://www.sciencedirect.com/science/article/pii/0017931081902209`.

Owen, J M and R H Rogers (1995). *Flow and Heat Transfer in Rotating Disc Systems, Volume 2: Rotating Cavities*. English. Research Studies Press.

Radenac, Emmanuel, Jérémie Gressier, and Pierre Millan (2014). “Methodology of numerical coupling for transient conjugate heat transfer”. In: *Computers & Fluids* 100, pp. 95–107.

Schindler, Alexander P. (2022). “Partitioned Modeling of Transient Conjugate Heat Transfer”. PhD thesis. Universität Stuttgart.

Schindler, Alexander P., Stefan Brack, and Jens von Wolfersdorf (2019). “Coupled FE-CFD analysis of transient conjugate heat transfer”. In: *13 th European Conference on Turbomachinery Fluid dynamics & Thermodynamics*. EUROPEAN TURBOMACHINERY SOCIETY.

Sun, Zixiang et al. (May 2011). “Coupled Aerothermomechanical Simulation for a Turbine Disk Through a Full Transient Cycle”. In: *Journal of Turbomachinery* 134.1. 011014. ISSN: 0889-504X. DOI: `10.1115/1.4003242`. eprint: `https://asmedigitalcollection.asme.org/turbomachinery/article-pdf/134/1/011014/5848876/011014\_1.pdf`. URL: `https://doi.org/10.1115/1.4003242`.