Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 126 (201709) 746–753

Energy Procedia

www.elsevier.com/locate/procedia

72nd Conference of the Italian Thermal Machines Engineering Association, ATI2017, 6–8 September 2017, Lecce, Italy

Finite element transient modelling for whole engine-secondary air system thermomechanical analysis

Sabrina Giuntini[a,*], Antonio Andreini[a], Giulio Cappuccini[a], Bruno Facchini[a]

[a] *Department of Industrial Engineering, University of Florence, Via Santa Marta 3, 50139, Firenze, Italy*

Abstract

This paper presents a new procedure developed in cooperation with Ansaldo Energia and aimed to predict metal temperatures in a gas turbine whole engine with an axisymmetric transient finite element approach. The 2D model includes a dedicated thermal fluid network where mass flow rates and pressure distributions are provided by external fluid network solvers in terms of time serie, while fluid-metal temperatures are computed through a customized version of CalculiX®. This work represents a first insight about a fully integrated WEM (*Whole Engine Modelling*) procedure currently under development. The future implementation steps will be oriented to the usage of a customized version of the native CalculiX®fluid network solver and the implementation of a system of monitoring and updating of the secondary air system (SAS) geometry. The aim is to progress from the current partly coupled approach with previously assessed mass flow and pressure distributions, to a fully integrated procedure able to take into account the interaction between the SAS fluid properties and the modifications in the geometry caused by mechanical and thermal loads.

In this paper, the methodology will be presented introducing some details about the main modelling aspects and illustrating some preliminary results from the test of the procedure applied to a simplified model representative of a real engine geometry under transient conditions.

Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association

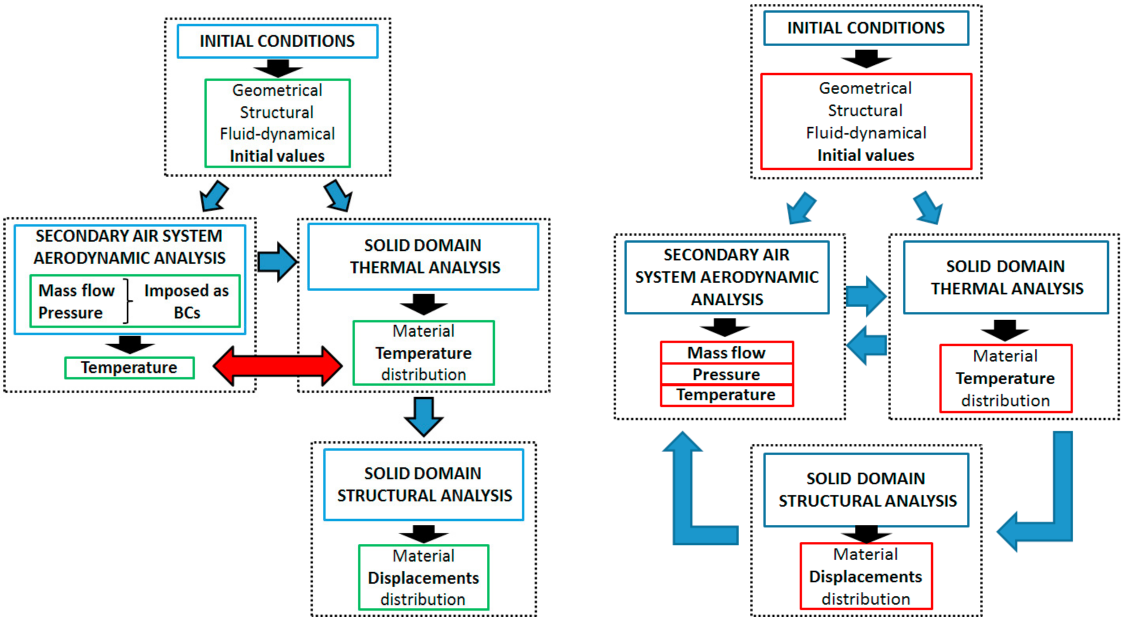
Keywords: whole engine modelling; gas turbine; heat transfer; partly coupled procedure; fully coupled procedure; CHT; fluid network solver

1. Introduction

Due to changes in power market in recent years, operation conditions of large heavy duty power generation turbines have been deeply modified. In order to manage the thermal and mechanical stresses encountered in their repeated transient operations, the investigation of the heat transfer between secondary air systems and structural components represents a decisive point in the design process.

* Corresponding author. Tel.:+39-055-275-8772 ; fax: +39-055-479-6342.
E-mail address: sabrina.giuntini@htc.de.unifi.it

1876-6102 © 2017 The Authors. Published by Elsevier Ltd.
Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association
10.1016/j.egypro.2017.08.231



(a) Partly coupled *WEM* approach. (b) Coupled Fully Integrated *WEM* approach.

Fig. 1. Coupling approaches.

In literature, despite sub-classifications sometimes not univocal, primarily two approaches are presented for the solution of the solid-fluid heat transfer problem. One is the fully conjugate, based on the solution of a unique domain through the application of one solver for the Navier Stokes and Fourier equations. A number of works show the application of the conjugate analysis to engine components, such as blades and vanes [1], [2], and rotor-stator systems [3]. The other main approach is based on the distinction of the different convective and conductive problems solved separately in a monolithic or partitioned manner [4] according to the method adopted for the integration of the solution of one domain in the other. In industries such kind of methodologies are generally chosen for their reduced computational costs and the possibility to carry out *whole engine modelling* (WEM) analyses in reasonable time. A coupling scheme example is the FEM-CFD application. It has been adopted for various analyses of components subjected to thermal transient cycle, such as pre-swirl systems [5], [6] and cavities [7], [8]. However the typical limitation of using CFD application is the computational cost. Consequently CFD tools find a collocation in a WEM process only as support to 1D simulations (which generally substitute the CFD analyses) for the investigation of those phenomena that one-dimensional models cannot catch. Besides containing computational costs, the other crucial point in the WEM process is accounting the strong coupling and non linearity in the heat transfer process during transient operations. During transient cycles indeed, components geometries can vary significantly according to the thermal gradients, modifying pressure losses and mass flow rates in the air system, affecting in turn flow and material temperatures [9]. In the aero-thermal FEM-CFD analyses the general approach is to account the effects of seal deformation by switching mass flow inlet and static pressure outlet boundary conditions between discrete levels of corresponding ramp points of the engine cycle [10], [11]. In [10] a fully integrated FEM-CFD aero-thermomechanical transient analysis through a flight cycle on an engine is presented. Anyhow due to the reduced computational costs, the WEM process is mostly oriented on iterative decoupled methods based on dimensionally heterogeneous models applying 1D fluid network solvers providing coolant flow rates and pressures to a 2D/3D finite element code for the solid domain [12], [4]. Industrial practices generally carry out the performance of fluid network solution, thermal analysis and solid deformation evaluation in an iterative separated way [9, 13]. Being based on a sequential approach this methodologies cannot catch the coupling nature of the phenomena. Nevertheless recently, procedures have been oriented towards coupling approach such as that presented in [4].

This paper presents a partially coupled procedure (Fig. 1a) aimed to predict heat loads on a gas turbine engine, developed in collaboration with Ansaldo Energia. The present work represents a first step of a procedure, still under development, aimed to carry out calculations on a whole engine model with a fully integrated approach (Fig. 1b). The current development step of the procedure is based on the solution of the whole engine structure from a point of view of the thermal and structural fields with the application of a customized version of the CalculiX®FEM solver. Fluid properties from the secondary air system in terms of mass flow split, pressure distribution and initial temperature field are provided to CalculiX®from a previous secondary air system (SAS) solution. Such analyses are performed with in-house one-dimensional fluid network solvers. In order to take into account the reciprocal interactions between the different phenomena involved in the transient operation of the engine, the integration of the native Calculix®fluid network solver is scheduled as a future step of the procedure development. In literature already other works based on CalculiX®are present and demonstrate the fluid network tool capabilities with satisfactory results [13]. The integration of the fluid network solver would allow to exploit completely the Calculix®modular features (SAS, thermal, structural solutions [9, 13]) to perform not only a coupled solution of the fluid and structure thermal fields but also to develop a method of updating and iterating on SAS geometries with a fully integrated approach. The future steps will include also the implementation of dedicated models for those features related to more specific aspects of gas turbine design.

The current procedure has been tested with a simplified test case representative of real engine typical configurations. Discussions about some fundamental modelling aspects will be presented with the preliminary results from a thermal transient analysis.

Nomenclature

A	Area $[m^2]$	Q	Heat flux rate $[W]$
C	Absolute velocity $[m\ s^{-1}]$	R	Radius $[m]$
C_p	Specific heat at constant $[-]$	SR	Swirl ratio $[-]$
HTC	Heat transfer coefficient $[W/m^2K]$	T	Temperature $[K]$
$\dot{m}$	Mass flow $[kg/s]$	U	Circumferential velocity $[m\ s^{-1}]$
Greeks			
ω	Rotational velocity	$[rad\ s^{-1}]$	
Subscripts and Superscripts			
abs	Absolute	k	k-th
add	Additional	out	Outlet
$conv$	Convective	rel	Relative
f	Fluid	t	Tangential
i	i-th	w	Wall
in	inlet	0	Total
j	j-th		

2. Procedure description

2.1. Fluid network solution

Being the fluid network solution imposed by external tools, the fluid network configuration adopted inside the CalculiX®model is the *purely thermal* one [14], through which mass flow rates and pressure distributions are fixed as boundary conditions and the only unknowns are the temperatures. Temperatures, where not imposed on the boundaries, are estimated solving the heat flux between metal and fluid with a fully coupling in temperature assessment between network and structural nodes since temperature unknowns are solved simultaneously.

2.2. Boundary conditions

In addition to secondary air flow splits and pressure distributions, HTC values on the secondary air system can be imposed as fixed values or can be calculated runtime. The same applies for the main flow loads whose fluid properties and HTC values can be obtained from experimental data or aerodynamic computations. As conventional industrial practice, all these quantities can be scaled and expressed as a function of time in order to reproduce the transient cycle conditions. The same convective coefficients can be expressed through implemented correlation as functions of significant parameters such as varying mass flow rates, fluid and wall temperatures. The possibility of catching the dependence of HTC values from the fluid properties is a crucial point, in order to reproduce a whole engine model running under real operating conditions.

2.3. FEM 2D Thermal conduction model

The solid domain is modelled with an axisymmetric approach with proper modelling for the treatment of 3D features. Axisymmetric approach has the great advantage of containing computational cost. Once the main flow and secondary air thermal loads are applied, convection, conduction and radiation problems are solved by the FEM module of CalculiX®v.2.11. Transient analyses are carried out to obtain component temperatures, displacements and stresses through a real engine cycle.

2.4. Iterative process

The main execution program for each time step discretizing the transient cycle, performs an iterative process in which the convergence in the coupled aero-thermomechanical fields is searched. The simulation starts with an initialization of the variable at the first instant, in general this is set to the cold engine conditions. Mass flow rates and pressures from the secondary air system and boundary temperatures are applied to the fluid network nodes and then are used for the formulation of convective loads and mechanical ones. Structure and fluid nodes temperatures are solved at once, in a one unique matrix. Thermal and stress fields are therefore obtained. The changes in the structure and network temperatures are assumed as new conditions for the convective solution for the next iteration.

3. Test case and calculation setting

A 2D axisymmetric gas turbine representative test case, with stator and rotor components, is considered. In particular it refers to a compressor arrangement with typical secondary air system elements such as rotating cavities, holes, air fulfilled rotating annulus, stator-rotor interface cavities and passages. As illustrated in (Fig. 2a) the mass flow rates come from three different inlets named A,B,C. Mass flow from inlet A represents air flowing in the main channel, mass flow from inlet B simulates an air contribution from the stator component to the rotor-stator cavity in which a percentage of air coming from rotor cavity merges.

3.1. CalculiX model setting

The CalculiX®model setting was based on the application of 313 boundary conditions, comprehending explicit convective conditions, forced convection with fluid elements, heat fluxes and boundary temperature, pressure and mass flow rates. The setup required a number of user coding operations to complete the features already present in the code and fulfill the proper modelling required by the test case. Below more detailed discussions about some fundamental modelling aspects are reported.

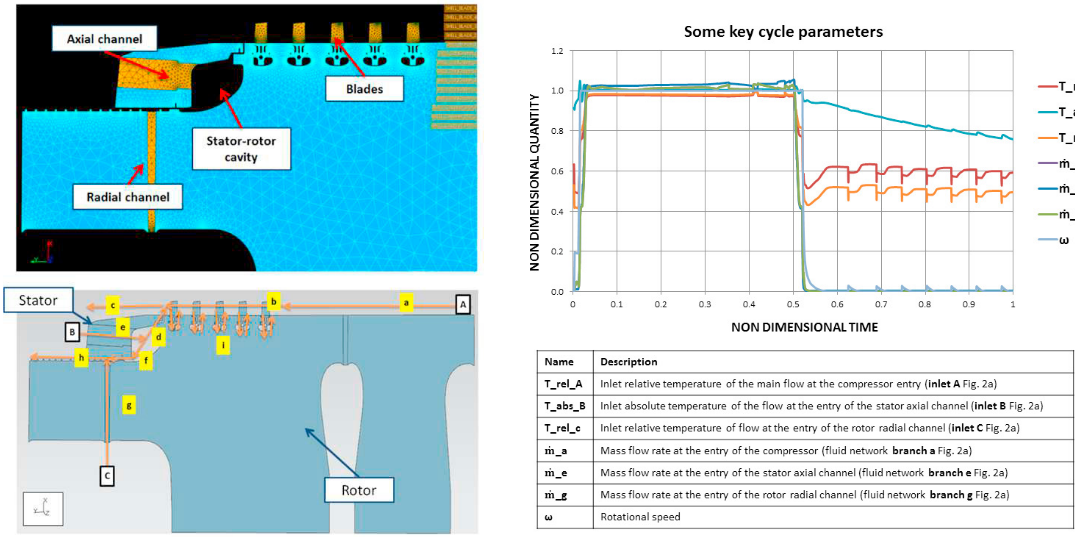
3.1.1. Rotational effects managing

Rotation introduces effects both on the solid and fluid domain. Concerning the metal, centrifugal distributed loading can be applied to rotating components. Regarding the action of rotation on the fluid domain, pumping effects affect flows along radial channel and cavities and they have been taken into account providing concentrated heat fluxes on the fluid nodes of each i-th fluid element involved in the phenomenon. Assuming a relative frame of reference, these heat fluxes are a function of the radius R through the temperature radial variation $\triangle T_{0,rel}(R)$, according to:

$$\dot{Q}_i = \dot{m} \cdot C_p \cdot \triangle T_{0,rel}(R) \tag{1}$$

The term $\triangle T_{0,rel}(R)$ for a rotating channel can be derived from the rothalpy conservation:

$$\triangle T_{0,rel,i} = \omega^2 \cdot \frac{\left(R^2_{out,i} - R^2_{in,i}\right)}{2 \cdot C_p} \tag{2}$$



Name	Description
T_rel_A	Inlet relative temperature of the main flow at the compressor entry (**inlet A** Fig. 2a)
T_abs_B	Inlet absolute temperature of the flow at the entry of the stator axial channel (**inlet B** Fig. 2a)
T_rel_c	Inlet relative temperature of flow at the entry of the rotor radial channel (**inlet C** Fig. 2a)
ṁ_a	Mass flow rate at the entry of the compressor (fluid network **branch a** Fig. 2a)
ṁ_e	Mass flow rate at the entry of the stator axial channel (fluid network **branch e** Fig. 2a)
ṁ_g	Mass flow rate at the entry of the rotor radial channel (fluid network **branch g** Fig. 2a)
ω	Rotational speed

(a) Test case 2D geometry and fluid network flows splits.

(b) Some cycle parameters: trends in time of rotational speed, temperatures and mass flow rates for the main ducts.

Fig. 2. Test case geometry, fluid network and key parameters.

Therefore assuming the heat transfer solution is carried out in a relative frame of reference, for the case of generalized pipes, if rotation occurs, the outlet node temperature will change according to the Eq. 2 where ω is the rotational speed and R_{in} and R_{out} are the radius of the inlet and outlet gate of the i-th element. To implement the temperature changes generated by the pumping effect, the artificial heat flux specified in Eq. 1 is added to the outlet node energy balance of the element. Similar considerations can be formulated for the change of reference system, necessity which can arise when fluid passes from an absolute reference to a relative one and vice-versa. The change in temperature:

$$T_{0,abs} - T_{0,rel} = -\frac{U^2 - 2 \cdot U \cdot C_t}{2 \cdot C_p} \tag{3}$$

is implemented through the same method based on the application of an artificial heat flux (Eq. 1) on the node modelling the reference frame passage at the considered radius.

An other important topic regarding rotational effects managing is the air flowing in the interface between statoric and rotoric components. The fluid network located between rotor and stator exchanges heat with both rotor and stator components at its own total temperature. Each node of this stream is characterized by only one value of total temperature. The issue is that the stator surface exchanges with the fluid at its own total absolute temperature ($T_{0,abs,fluid}$), while the rotor, due to its rotation, exchanges with the fluid at its own total relative temperature ($T_{0,rel,fluid}$). Considering the equation of the convective heat transfer written in a reference frame build-in with the rotor (applying therefore the fluid relative temperature $T_{0,rel,fluid}$), and handling the equation summing and subtracting the absolute temperature $T_{0,abs,fluid}$, we obtain:

$$\dot{Q} = HTC \cdot A \cdot (T_w - T_{0,abs,f} + T_{0,abs,f} - T_{0,rel,f}) \tag{4}$$

Considering the stream in the absolute frame of reference, substituting the Eq. 3 in Eq. 4 and considering the definition of the swirl ratio ($SR = \frac{C_t}{U}$), a correction in terms of an additional distributed heat flux is applied to the rotor side on the rotor elements facing the fluid in presence of a radius variation, and with opposite sign on the fluid nodes of the stream:

$$\dot{Q}_{conv,add} = HTC_k \cdot A_k \cdot (1 - 2 \cdot SR_j) \cdot \left(\frac{(\omega \cdot R_{f,j})^2}{2 \cdot C_p} \right) \tag{5}$$

3.1.2. Axisymmetric-plane stress elements coupling and HTC scaling

Stator and rotor components were modeled as axisymmetric exception for the zones of the mesh corresponding to channels and blades. In these cases the element type applied was the plane stress with appropriate modelling assumptions to take account of 3D features and axisymmetric-plane stress elements coupling in terms of conduction. This followed the conventional industrial practice with features such as bolts, holes, etc. modeled using an equivalent wall thickness and proper HTC values scaling. In particular CalculiX®uses on the 2D axisymmetric model solution an expansion angle fixed at 2°and plane element thickness is scaled automatically by the solver in order to consider the proper fraction of material on the expansion angle of 2°. Heat transfer boundary have been evaluated therefore considering the 2°expansion (and the consequent thickness scaling), the eventual radial shaping of material thickness and the typology of surfaces on which the HTCs were applied, i.e. if developed from axisymmetric or plane elements.

3.1.3. Rotating closed cavities handling

Generally, rotor structure is characterized by the presence of manifold cavities. Some of them are completely closed, not fed by secondary air system branches. In these entirely closed gas-filled rotating annulus, the predominant convective phenomenon is the natural convection [15]. In the central rotor cavity of the model, in order to simulate the presence of enclosed thermal masses that exchange heat through natural convection with the metal surfaces, a disjointed fluid element with a very small flow rate at its inlet, has been adopted. The flow rate very small (not actually simulating an efflux but the very low thermal inertia) will guarantee that ultimately the outlet temperature becomes an average of those of the solid walls, independently from the conditions imposed at its inlet. Concerning radiative effects, in the case of closed cavity, view factors can be scaled to one exactly and the sink temperature can be calculated by the solver based on the interaction of the surface at stake with all other cavity radiation surfaces.

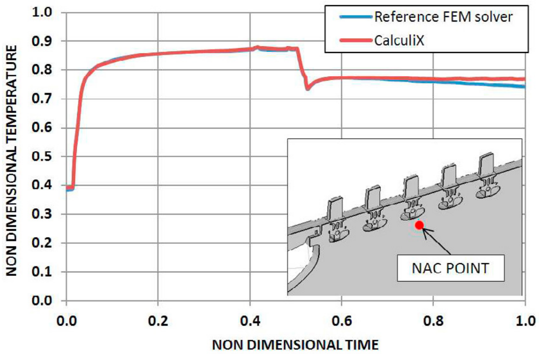
4. Discussion of results

The 2D axisimmetric model was discretized with 17973 linear triangular elements with 10318 nodes. It has been run with the presented procedure based on CalculiX®and with a reference FEM solver. Some points in the solid domain have been monitored in time and comparisons have been made in order to have a preliminary assessment of the consistency of the modelling adopted in the procedure.

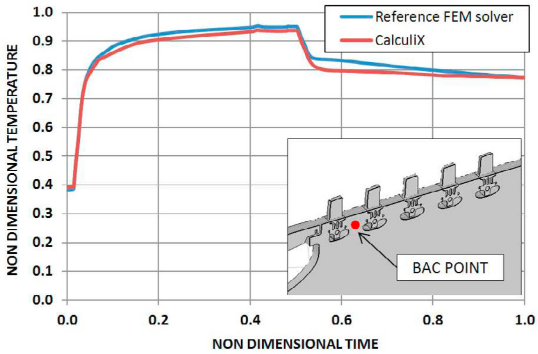
In both cases the model setup follows the same guidelines in terms of boundary conditions, loads and modeled phenomena. The transient cycle chosen was inspired to real engine operating conditions with the startup ramp, regime operation and the shutdown phase. The transient cycle was simulated for a time duration of about 70000 s with the same condition sequence for both setups in terms of time points discretizing the complete extent of the simulation and fixing the quantities values during the transient cycle.

The procedure demonstrated to be able to follow the transient operation, catching a coherent behavior of the thermal model according to the engine cycle and the physic of the problem. As shown in Fig. 3, points closer to the main flow are subjected to sudden rise in temperature in the startup phase and to higher temperature levels in general. Similarly, due to their exposition to the main flow, the shutdown phase is characterized by a quick temperature reduction during the firing cut off. During this phase temperatures across main channel, rotor speed and mass flow drops significantly (Fig. 2b). Since the mass flow entering into the turbine region is very small a slight increase in temperatures across components stations is generally observed due to heat soak effect, phenomenon causing fluid temperature and pressures increase due to heat received from metal, after the engine is turned off. The effect of the rotor thermal inertia also affects components like blades which are involved in a temporary increase of the temperature: while flows are quite exhausted the predominant phenomenon is the conduction of the heat accumulated in the rotor till that moment as it can be observed around the halfway point of the cycle in Fig. 3a and 3c. After that phase, a gradual decrease in temperatures follows. Other points like those located in zone fed by cooling and purge flows (Fig. 3d and 3e) or those located in more internal zone of the rotor (Fig. 3f) where the thermal inertia is higher, experience more moderately the above cited temperature variations, showing smoother trends.

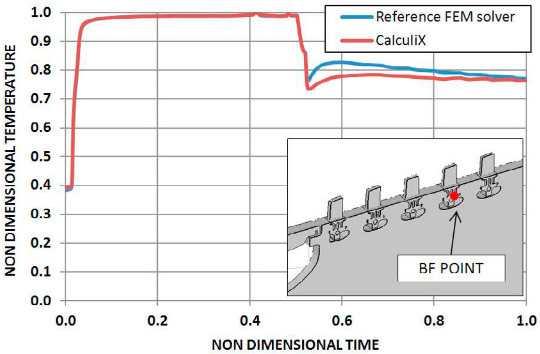
As reported in Fig. 3 the agreement with the prediction of the reference FEM solver is generally good and differences not exceed the 6.5 %. Reported temperatures are scaled with respect to a reference temperature T_{ref} (approximately the maximum value detected on the observed points). Observed differences are probably due to a not fully equivalent level of discretization in the two model settings. In CalculiX®the fluid network setup has been led by considerations about the spatial discretization of the solid domain and the expected complexity of the heat transfer. In the reference FEM solver the fluid network definition has been carried out by an automatic algorithm of discretization which can differ from that set in CalculiX®. Fluid properties generally are evaluated at a film temperature defined



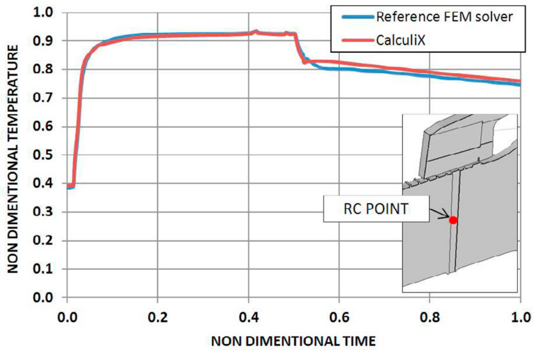
(a) Near anchoring cavity thermal response.



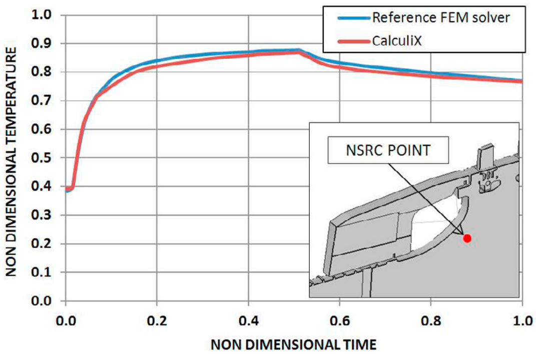
(b) Between anchoring cavities thermal response.



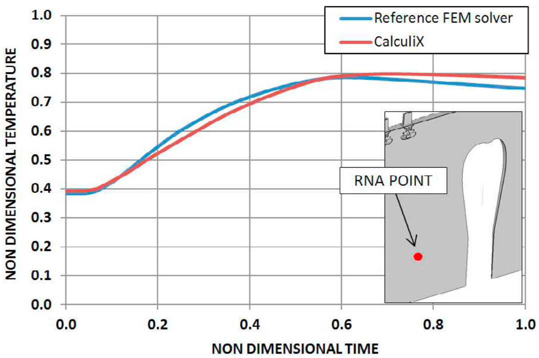
(c) Blade foot thermal response.



(d) Radial channel thermal response.



(e) Near stator-rotor cavity thermal response - Rotor side.



(f) Rotor near axis thermal response.

Fig. 3. Non dimensional temperature trend versus non dimensional time at some solution locations.

by the mean of the flow and wall temperature, while in CalculiX®properties are evaluated at the fluid temperature. For non-axisymmetric features like blades and holes, convective loads on plane elements and at the interfaces of the latter with the axisymmetric ones have been applied in CalculiX®with a proper scaling of the heat transfer coefficient. The scaling is based on the possibility of loading separately both kinds of elements, and not only on the front and rear faces (planes) but also at the interface. Here the axisymmetric elements have a surface of exchange developed on the 360°circumferential expansion and the plane ones have the specified thickness. HTC scaling takes into account the typology of element on which it is applied and therefore the corresponding surface of exchange. In the reference FEM solver instead the definition of the load is made on the geometric edge representing the boundary between the axisymmetric and the plane domains, making the actual exchange surface used by the solver of difficult definition.

Such kind of setup misalignments can justify the differences in predictions, considered however acceptable for the goal of these preliminary comparisons aimed at a first insight in the solver capabilities and consistency of the non native modelling features introduced.

Conclusions

Transient thermal evaluations are fundamental to estimate the clearances and stator-rotor gaps, and predict the stress induced in the engine components. The procedure discussed here aims at investigating the heat transfer process during a transient thermal cycle of the whole engine. The methodology is based on a FEM approach through a customized version of CalculiX®. Up to now the process is partly coupled since only the fluid-thermal and thermal-structural couplings are accounted for, without an integrated updating of the displacements on the secondary air system geometry, unless it is performed manually by the user. The procedure has been tested with a simplified test case representative of real engine geometries. The calculation has been carried out on a transient operating cycle. Results have been preliminary compared with results of a reference FEM solver, showing good agreement.

Acknowledgements

The authors would like to acknowledge Ansaldo Energia for the support and cooperation in the presented project and Dr. G. Dhondt from MTU Aero Engines Munich for his precious advices.

References

[1] Bohn, D., Ren, J., Kusterer, K.. Conjugate Heat Transfer Analysis for Film Cooling Configurations With Different Hole Geometries. Proceedings of ASME Turbo Expo 2003: Power for Land, Sea, and Air 2003;Paper No. GT2003-38369.

[2] Lin, G., Kusterer, K., Ayed, A.H., Bohn, D., Sugimoto, T.. Conjugate Heat Transfer Analysis of Convection-cooled Turbine Vanes Using γ-Reθ Transition Model. International Journal of Gas Turbine, Propulsion and Power Systems 2014;6(3).

[3] Okita, Y., Yamawaki, S.. Conjugate Heat Transfer Analysis of Turbine Rotor-Stator System. Proceedings of ASME Turbo Expo 2002: Power for Land, Sea, and Air 2002;Paper No.GT2002-30615.

[4] Ganine, V., Hills, N., Miller, M., Barnes, C., Curzons, S., Turner, L., et al. Implicit Heterogeneous 1D/2D Coupling for Aero-Thermo-Mechanical Simulation of Secondary Air Systems. Proceedings of ASME Turbo Expo 2015: Turbine Technical Conference and Exposition 2015;Paper No. GT2015-43406.

[5] Illingworth, J.B., Hills, N.J., Barnes, C.J.. 3D Fluid-Solid Heat Transfer Coupling of an Aero Engine Pre-Swirl System. Proceedings of ASME Turbo Expo 2005: Power for Land, Sea, and Air 2005;Paper No. GT2005-68939.

[6] Javiya, U., Chew, J., Hills, N., Dullenkopf, K., Scanlon, T.. Validation of CFD and Coupled Fluid-Solid Modelling for a Direct Transfer Pre-Swirl System. Proceedings of ASME Turbo Expo 2012: Turbine Technical Conference and Exposition 2012;Paper No. GT2012-69056.

[7] Kumar, B.V., Chew, J.W., Hills, N.J.. Rotating Flow and Heat Transfer in Cylindrical Cavities With Radial Inflow. ASME Journal of Engineering for Gas Turbines and Power 2013;135(3):032502.

[8] May, D., Chew, J.W., Scanlon, T.J.. Prediction of Deswirled Radial Inflow in Rotating Cavities With Hysteresis. ASME Journal of Turbomachinery 2013;135(4):041025.

[9] Muller, Y.. Secondary Air System Model for Integrated Thermomechanical Analysis of a Jet Engine. Proceedings of ASME Turbo Expo 2008: Power for Land, Sea, and Air 2008;Paper No. GT2008-50078.

[10] Ganine, V., Amirante, D., Hills, N.J.. Aero-Thermo-Mechanical Modelling and Validation of Transient Effects in a High Pressure Turbine Internal Air System. Proceedings of ASME Turbo Expo 2016: Power for Land, Sea 2016;Paper No. GT2016-57739.

[11] Sun, Z., Chew, J.W., Hills, N.J., Volkov, K.N., Barnes, C.J.. Efficient Finite Element Analysis/Computational Fluid Dynamics Thermal Coupling for Engineering Applications. ASME Journal of turbomachinery 2010;132(3):031016.

[12] Peschiulli, A., Coutandin, D., Del Cioppo, M., Damasio, M.. Development of a Numerical Procedure for Integrated Multidisciplinary Thermal-Fluid-Structural Analysis of an Aeroengine Turbine. Proceedings of ASME Turbo Expo 2009: Power for Land, Sea, and Air 2009;Paper No. GT2009-59875.

[13] Muller, Y.. Integrated Fluid Network-Thermomechanical Approach for the Coupled Analysis of a Jet Engine. Proceedings of ASME Turbo Expo 2009: Power for Land, Sea, and Air 2009;Paper No. GT2009-59104.

[14] Dhondt, G.. Calculix crunchix user's manual version 2.11 2016;.

[15] Bohn, D., Gier, J.. The effect of turbulence on the heat transfer in closed gas-filled rotating annuli. ASME Journal of Engineering for Gas Turbines and Power 1998;120(4):824–830.