ASME TURBO EXPO 2009
Power for Land, Sea & Air
June,8-12, 2009 – Orlando, USA

GT2009- 59875

DEVELOPMENT OF A NUMERICAL PROCEDURE FOR INTEGRATED MULTIDISCIPLINARY THERMAL-FLUID-STRUCTURAL ANALISYS OF AN AEROENGINE TURBINE

Antonio Peschiulli*
Department of Aerospace Engineering
of Politecnico di Torino,
Corso Duca degli Abruzzi, 24, Torino,
Italy
e-mail: antonio.peschiulli@polito.it

Daniele Coutandin
Marco Del Cioppo
Avio S.p.A.
Via I maggio 108, 10040
Rivalta di Torino, Torino, Italy
e-mail:
Daniele.Coutandin@aviogroup.com
Marco.Delcioppo@aviogroup.com

Massimo Damasio
Exemplar S.r.l
C.so Castelfidardo 30/A,
Torino, Italy
e-mail:
massimo.damasio@exemplarsolutions.it

ABSTRACT

The aim of the work is to define a methodology and to develop specific tools that allow engineers to investigate and full characterize the thermal performance of an aero-engine turbine module.

The thermal behaviours of a complex system are the result of strong interactions between the fluid-dynamics aspects, the heat balance within each component and the geometric variations mainly due to the thermal and mechanical loads. All thoose phenomena are strictly connected and cannot be studied separately without introducing approximations and /or errors in the final results.

For long time the industrial approach in turbine design has been based on separate analyses of the phenomena with manual iterations between them to allowed acceptable solutions to be reached.

The new requirements in reducing the product time-to-market together with the need of higher accuracy in the design, have driven the development of new approaches based on the multi-disciplinary analysis integration.

This paper will summarise the AVIO approach to the turbine design procedure upgrade, mainly focused on the thermal analysis and clearances control.

A detail of the methodology used will be presented together with a description of the tools developed. A comparison between numerical predictions and experimental data (full engine test) will be reported.

INTRODUCTION

The strong incentive to improve the aero-engine performance in recent years, mainly connected to potential specific fuel consumption reduction, has led to optimizations of individual aspects of the engine system .One of the areas considered more strategic to guarantee a higher level of efficiency is the control of the clearances in turbine modules (Figure 1 and Figure 2): this means to be able to control the distances between rotating parts (mainly tip of blades) and static parts (mainly shrouds) in any engine operative conditions. Minimising this distance is the way to increase the turbine efficiency [ref.1]. Those gaps are the results of a lot of component deformations (blades elongation, disks thermal expansion, static parts displacement due to controlled cooling …), so they can be evaluated only if the thermal behaviour of each component can be well simulated and calculated. However to determine temperature distribution in each part of the turbine requires a knowledge of the fluid-dynamic characteristics of both hot and cold air streams (mainly mass flows with their pressures and temperatures in every passages of the turbines). Finally those fluid data can be calculated only if the geometry of air passages / gaps (mainly in seals region) are known, but, as mentioned above, they are a function of the deformation of the parts. This loop of physical interactions must be taken into account if the quality of the results are to satisfy the really strict requirements described above.

Without considering this approach the final temperature distributions of the components must also be affected by higher uncertainty, that means the requires the introduction of leveles of margin in the design, both in life prediction and in air consumption (for cooling and purge). The required performances

* Author of correspondence

of the new generation of engines will not allow such kind of margins.

In the following paragraphs a method to improve the thermal evaluation of a turbine module, using a multidisciplinary approach, will be described. The method has been implemented in a tool that will be also described, and some predicted results will be compared with available engine data.

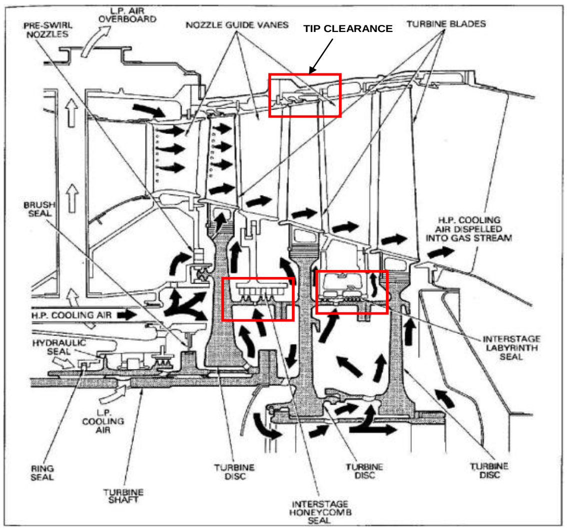


Figure 1*: Turbine typical layout with schematic flow network [ref.1]*

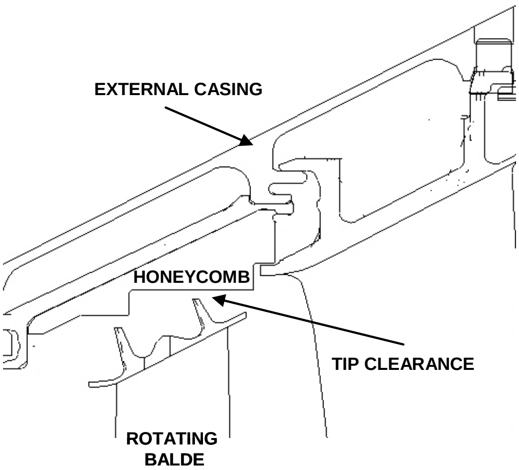


Figure 2*: Detail of a typical tip clearance geometry*

MULTIDISCIPLINARY APPROACHES

An evidence of the increasing in interest of the technical institutions on integration problems can be found in the creation of official committees dedicated to this aspect. It is one of these committees, the AIAA MDO Technical Committee [ref. 1], that in 1991 has provided a referenced definition of the "Multi-Disciplinary Analysis":i.e.

> *"A methodology for the design of complex engineering system and subsystem that coherently exploits the synergism of mutually interacting phenomena"*

Starting from this definition, it is understandable how improving design complexity can not be unaffected by the mutual interactions between physical phenomena that occur in a gas turbine. Moreover the accurate of analysis is improved as the input and the output of different analyses are connected to each other.

In particular, the value of in diffusing this kind of multidisciplinary approach, can be recognize in the common benefits that those investigations can have on the final products in the areas of:

- Quality of the design process;
- Reliability and Robust Control;

The first aspect relates to the enhancement of quality both during the design phase and in the final product. The benefit of increase quality during the design phase is achieved by customizing the usual procedure that are required to perform the analysis. Integrated codes enable the to convertion of outputs that comes from one process to another process, deleting the inefficiency due to transcription of data from one source to others. Moreover the quality of the process is also affected because it is possible to control performance parameter of different design processes to optimize product requirements or to achieve specific technical targets. Generally speaking the integration of the different design area of development can be considered as an approach to the quality standard of the design, also known as Design For Six Sigma (DFSS).

The second point relates to the chance of building, with the aim of integrating tools, for analysis of reliability of robust control whenever robustness is defined as a specific target of the design. The integrated environment allows investigation by means of high number of simulations,the response of the system to, for example, inaccuracies of the manufacturing process or uncertainness in design parameters, and then to select the more promising solution.

The situation described of the aero-thermal-deformation analysis in a turbine, can be treated as a classical example of the complexity of non-hierarchical problems. Those cases are known to be problems where the possible solutions can not be found by solving a series of subsequent subcases or subspaces, but it is the result of the iteration between each of the previous subspaces. In a non-hierarchical case, the process – defined as the sequence of operations that must be performed to achieve a final resolution – becomes recurrent because the input of specific analysis are the result of following analysis outputs (see *Figure 3*), hence loops of design phases are needed in achieving a final and consistent solution.

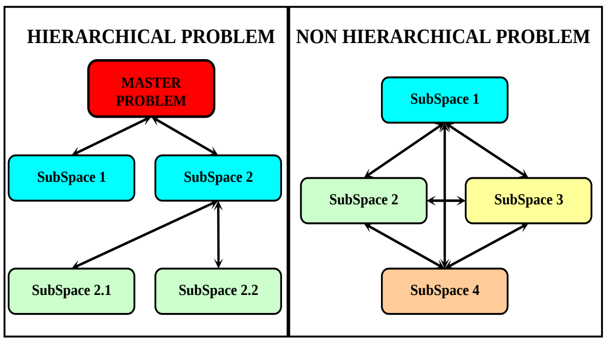


Figure 3*: Hierarchical and non-hierarchical flowcharts*

In order to solve a non-hierarchical problem, different strategies may be applied, and the choice between which one of them is driven by the gain in efficiency,i.e. the reduction in the time consumed in performing such specific simulations.

Moreover recent efforts in coupling non-hierarchical problem with optimization problems, offers a possible new approach to the solution of this kind of analysis. It is possible to summarize the different strategies above in two main families [ref. 3]:

- Multi Design Feasible;

- Optimization coupled integrated analysis.

In the fist case, the solution to the overall problem is obtained by reaching the solution in each subspace coupling output and input of the analysis, and controlling the residuum behaviours until convergence is achieved. In this case, for every loop of variables that are used as input and output of the different analyses, a loop between codes is required. Hence time consumption increases with increased complexity, measured by the number of coupled variables.

In the second case, every solver involved in the integrated simulation is un-coupled by creating an additional variable, which controls the difference between the hypothetical values used as input, and the output for the coupled codes. In this case, an external controller – the optimizer – is responsible of minimizing the inconsistencies between the coupled variables used by the solvers, and in this way, the feasible solution to the overall problem is obtained only at the end of the optimization.

According to Hulme [14], a method that decreases the usage of iterative solutions would be surely preferred in the case where solvers are represented by very timely design tools, like, in this case, two FEM formulations and a network solver. But, on the other hand, saving time consumed increases the risk of having less precision in the residuum balance of the coupled variables, especially when the number of these parameters is high. This has moved the direction of research to a different way for increasing the efficiency of the integration strategy.

The solution found has been studied for the specific design intent of the realized integration, and it is related to the particular physical phenomena that are encountered during the simulation. For this reason, it can not be disassociated from the integration problem presented, but it offers an example of method that can be adapted to different cases after a similar preliminary study.

Hereafter, before presenting the method, the design problem is described in order to explain how to apply the proposed integrated solution, and to introduce to the following test case.

NUMERICAL PROCESS

As mentioned before, the numerical process involved in this analysis refers to the full transient thermal analysis of an aero-engine turbine. The main features, or sub-process, of the main design flux can be summarised as:

- *Thermal analysis*, is the process that evaluates heat loads and temperature distributions in each region, and for each required time step. Due to complex geometries the problem needs to be solved using a Finite Element Method (FEM) approach, but, in general it can be simplified, during the design phase, as an axisymmetric problem. The thermal solver will find a solution (in steady or transient condition) by performing thermal balance through conduction, convection, and radiation heat exchanges. For doing this, it needs to know the mass flow distribution and characteristics near all the wetted surfaces.

- *Fluid-dynamic analysis* (SAS) is the process that allows the evaluation of the mass flow distributions within the secondary air system. Also in this case, the complexity of the network and geometries leads to the impossibility of using full 3D "Computational Fluid Dynamic (CFD) approach". The solution is to follow a flow-network solver methodology, in which the fluid domain can be simulated by using a series of 1D models, each of them having dedicated correlations for simulated flow evaluation. The process requires as input the geometry (in particular values of seal gaps and other clearances), and a thermal map.

- *Deformation analysis*, is the process for the evaluation of the component deformation,i.e. the new flow path geometry. Displacements can be influenced by mechanical and thermal loads (thermal maps), so it is necessary to use a 2D axisymmetric model in order to calculate them correctly.

For a long time such simulations on complex systems, have been managed by considering a sequential calculation approach (Figure 4). Fluid and deformation analyses were in general done only in a stabilized condition, and data interpolated during the thermal transient of a whole engine mission. Some loops were necessary in order to have coherent results between the three processes. In any case the final uncertainty introduced, cannot be avoided just by increasing the number of loops.

The new requirements of engine clearances and component lifing controls can be translated into a thermal requirement, of having calculated temperatures with an uncertainty that must not exceed a standard deviation of about 10 degrees Celsius in the normal Low Pressure Turbine applications.

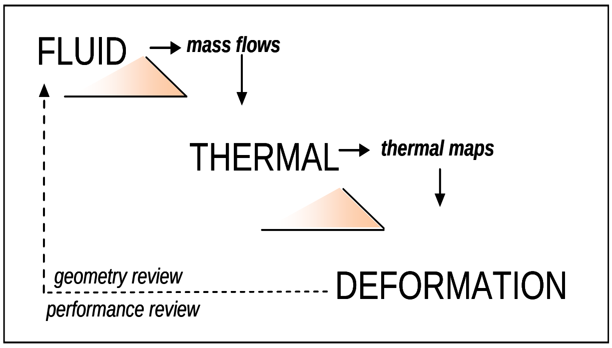


Figure 4: Sequential Calculation Approach

To achieve this goal it is necessary to change the design methodology and to move from a "*sequential thermal calculation approach*" to a "*full integrated thermal approach*".(Figure 5)

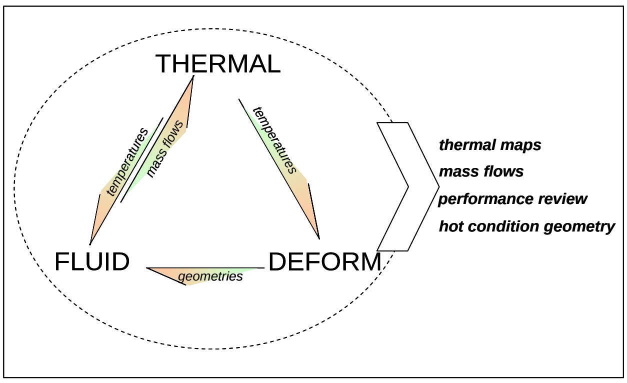


Figure 5*: Fully Integrated Thermal Approach*

This means takeing into account, at each time step of the analysis and at each convergence step, the complete status of the system from a thermal, fluid, and deformation point of view. In doing that each physical phenomena can be considered and evaluated with the exact boundary conditions determined by all the three aspects involved.

DEVELOPMENT OF THE INTEGRATED TOOL

The main idea of the present work was not to change the specific design practices available for each analysis, but to introduce all of them in an integrated multidisciplinary approach, as described above. The choice was then to use the well tested commercial codes already in-use, in stand alone mode, in a sequential approach. They are:

- MSC P-Thermal [ref. 11] for the heat balance. The code is widely used for this kind of analysis thanks also to the possible use of User Defined Libraries in which special users practices can be implemented. The same libraries have been used for the integration of the whole integrated procedure manager.

- FLOWMASTER [ref. 12] for flow network solver, with an extension for specific sub-cases phenomena simulation.

- MSC MARC for deformation analyses. The choice of this code has been mainly based on the high integration level already available with the thermal solver. Both usePATRAN as pre-processor, where models and loads can easily and automatically translate from one analisis to the other.

The integration have been then implemented at two levels:

- Graphical User Interface (GUI) integration;
- Simulating Codes & algorithms Integration.

1) Grafic User's Interface

The three analyses are in fact based on the three separated models (thermal, flow network and structural) but it is necessary to assure their coherence from the geometric and boundary conditions points of view. Two interfaces are available: PATRAN for the thermal and deformation solvers and FLOWMASTER GUI for the network solver. Also the user requirement to introduce only once the data that are used in more then one model became essential, if the final goal is the robustness and quality of the design procedure. This means to introduce the capability to translate the data from one model to the others. The choice has been made to use the thermal model prepared in PATRAN as the main one, and use it for translating the information directly to the others; this means essentially the two following steps:

a. convert and export the "advective" fluid network defined in the thermal model (P-THERMAL) into the Fluid-Dynamic Network of FLOWMASTER

b. convert and export the complete thermal model (P-THERMAL) into the deformation model (MARC)

Concerning a), dedicated objects have been created in PATRAN and associated to the beam elements simulating the flow network. Objects can be recognised by the FLOWMASTER GUI as typical pressure loss models (pipes, junctions, pressure source, orifice …) and the same network can automatically re-build. Some special objects need to be created (named TNODE) in order to transfer the temperature from the P-THERMAL node to the FLOWMASTER network.

Concerning b), geometry and mesh are automatically translated. Also some features of the mechanical loads and contacts can be automatically prepared in the deformation model based on some characteristics of the thermal model.

The process for the models preparation then became:

- to prepare the thermal model (that can be used also as stand alone)
- to translate the fluid network into FLOWMASTER and add the input data required.
- To translate the whole thermal model into the structural one (all done in PATRAN) and add the additional required inputs.

The high automation level introduced helps the user to avoid errors and to speed up this phase of the design process.

2) Simulating codes & algorithms integration

The FLUITHEST integration algorithm has been built taking into account the required flexibility to use a full integration (Thermal-Fluid-Deform) or only partial (Thermal-Fluid integration)

The algorithm type used for all the levels of integration is based on the technique of the Fixed Point Iteration, hereafter called as FPI. This technique has then been modifed in order to be adaptive and more efficient for the specific application presented here.

The FPI is based on the iterative loops of input and output variables coming from different codes, where it is required that the same variables, at the end of the simulations, will satisfy both calculation domains in a consistent way.

To show how this process works, a flow chart of the FPI methodology applied between only two different codes is presented in Figure 6.

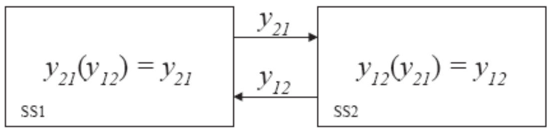


Figure 6 FPI process example applied between two codes

The boxes called SS1 and SS2 (SS is used to denote Sub System) can be two generic processes, in our case, a simulation tool with its specific code. Variable y21 is the generic output of SS1 that will be used as input by SS2. The same nomenclature is used for the other coupled variable y12.

The application of the classical method as in this example, will consist of the following calculation steps:

Step 1) initialization of one coupled variable (y^0_{12}); set iteration counter variable i = 0;
Step 2) next iteration : i = i +1;
Step 3) $y^i_{21} = y_{21}(y^{i-1}_{12})$
Step 4) $y^i_{12} = y_{12}(y^i_{21})$
Step 5) if $|y^i_{12} - y^{i-1}_{12}| < \varepsilon$ stop, else go to (Step 1).

The complete integration performed in FLUITHEST uses the same procedure presented above, but it is more complex and it involves the four codes. The loops of the coupled variables that are involved in the analysis are shown in Figure 7 using the notation known with the name of Design Structure Matrix – DSM [ref. 7].

Each link of the DSM indicates that the variables are coupled between the indicated solver. Hence for each line, based on the complexity of the simulated system, a set of parameters migrates automatically from one code to the other.

At the first tentative development of the simple FPI process, the integration has revealed the requirement of an high number of recurrences between codes and hence a high time spent.

The first modification introduced in the integration is represented by a limitation in the recurrences, as proposed and described in previous papers (see references [ref. 6] and [ref. 7]). This method is called "suspension", and it is basically an interruption of the coupled variables applied when the impact on the integrated simulation does not determine a significant variation on the convergence.

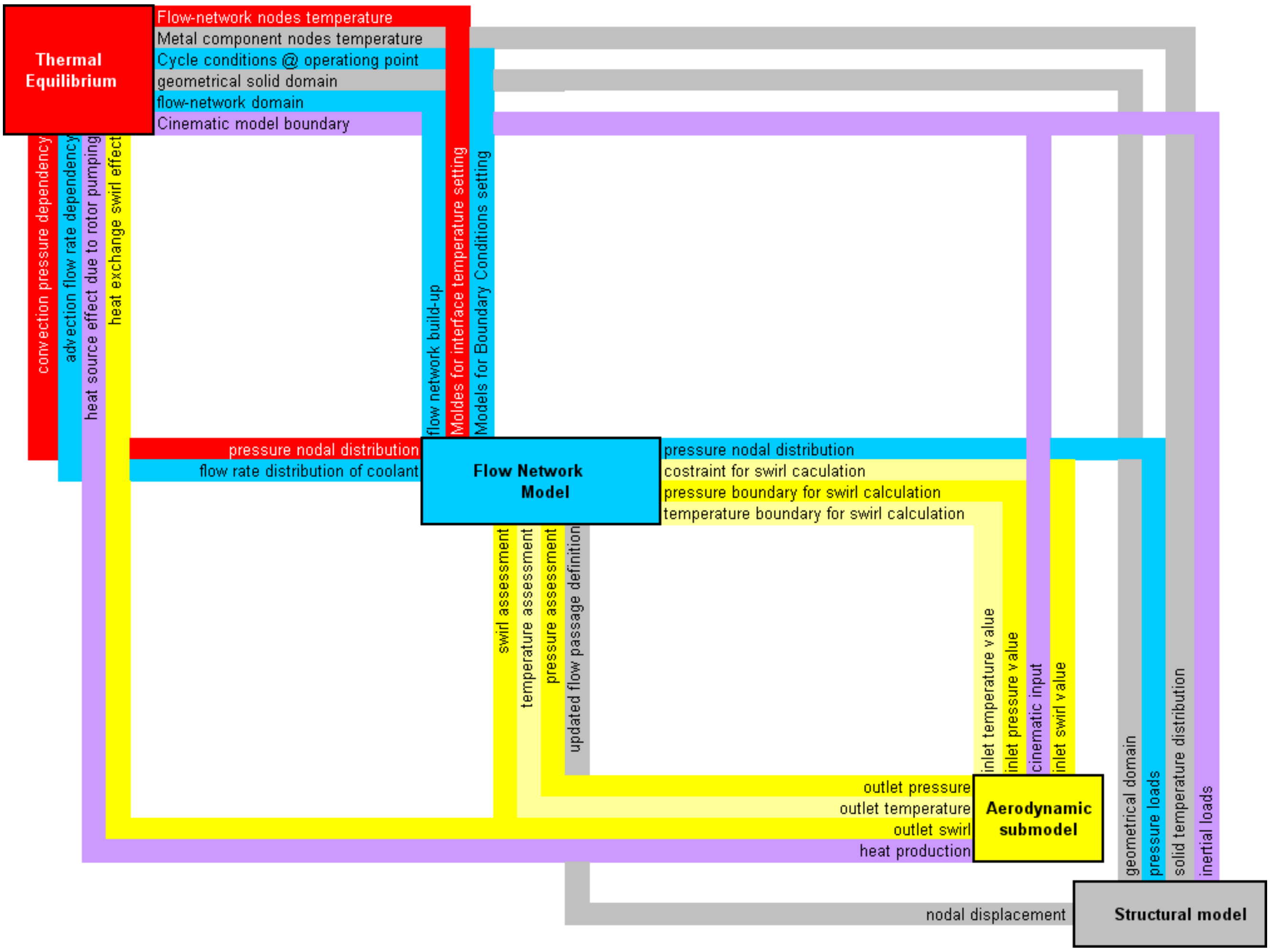


Figure 7 Matrix of the coupled variables used by the integrated codes as in the FLUITHEST process.

The formulation of the suspension criteria has been developed according to the results of the first test case presented hereafter, and it based on the analysis in the evolution of the coupled variables. Taking into consideration, for example, the evolution of the Thermal model and of the linked flow-network model, it was compared the maximum variation of temperature of the FE thermal model, to an index of efficiency for the variation of the mass-flow rate values. This index of efficiency was formulated as the number of efficient updated coupled variables of the network (considering the updating was efficient when the variation was higher than a tolerance value) divided by the number of coupled variables. Hence, an efficiency equal to one means that all the coupled variables are updated in the flow network, and therefore the integrated solver is far away from convergence, and from its final solution. In this situation, the suspension of the couplings variables can be applied.

Moreover, comparing the value of efficiency with the maximum value of variation in the nodal temperature of the Thermal model (it is worth a reminder that the nodal temperature is one of the key parameters that determine the variation in the flow network model results), it is also possible to control the phases in the integrated simulation, when coupled variable suspension is possible .

For example in Figure 8 it is shown that the efficiency is constantly equal from the first iteration up to iteration 600, then it decreases due to tolerances in the nodal temperature variation beign lower than 0.01 K. Then suspension is applied up to iteration 600 to speed up the process of convergence in order to obtain less results during this calculation phase and increase precision only at the end of the iterations.

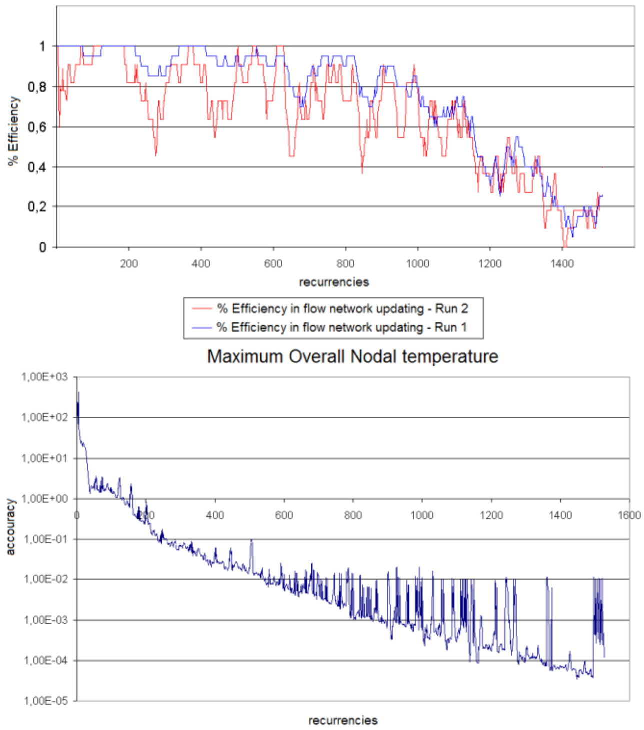


Figure 8 Evolution of efficiency and temperature variation for a test case.

Anyway, it is not easy to evaluate the effect in time reduction for any simulation, because it is highly sensitive to the model complexity, and the coupling of the overall system.

The second modification is the management of the accuracy required for each analysis during the integration . An analysis of the time consumed by each code during the iterations has highlighted that the mainly part of the time (90% of the time for recurrence), is required by the thermal solver (see Figure 9) . For that reason, the idea was to decrease the thermal solver accuracy during the first part of the simulation in order to reduce the overall time of the simulation. In the second part of Figure 9 it is shown that the benefit in terms of overall run time is a reduction of nearly 50% compared with in the simple FPI implementation. This modification does not affect the final result because the accuracy of the thermal solver is increased in the final steps of the integration up to the same value as in the first implementation.

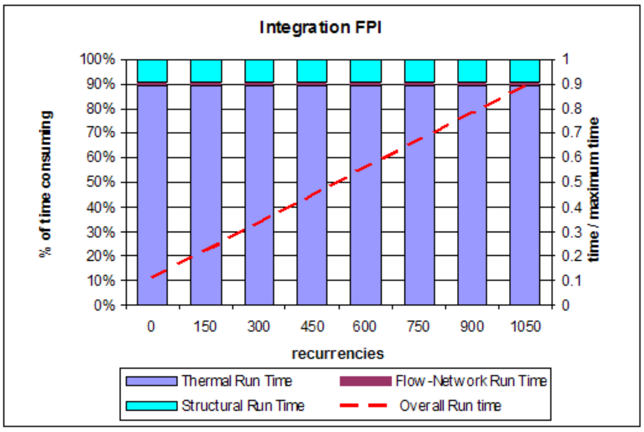
To summarize, the modifications introduced in the original version of the algorithm are:

a. to control the accuracy of the results of the single code during the convergence;
b. to control the number of iterations based on the development of the solutions themselves.

The above criteria has been implemented into the thermal solver code according to manual user guide instructions [ref. 11] , in order to make as simple as possible, the procedure for the starting of the simulation.

During Transient analysis the same criteria as steady state is used, but the algorithm monitors the delta temperature error during integration time step. Moreover, other parameters are checked to activate the fluid dynamic run: in particular all time dependent boundary conditions are controlled and compared with a convergence tolerance on loads. If loads don't change between two consecutive time steps no fluid-dynamic re-calculation is needed and only delta temperature controls the fluid dynamic runs.

The algorithm remains the same as described above, but in the case of the structural code, present in the loop, this also takes into account the clearance deflection convergence. This means that the variation in the percentage of monitored clearances is compared with a tolerance value on clearance convergence set by the user. Again, as in steady state the control is activated between steady state iterations, in transient analysis the control is applied also between integration time steps



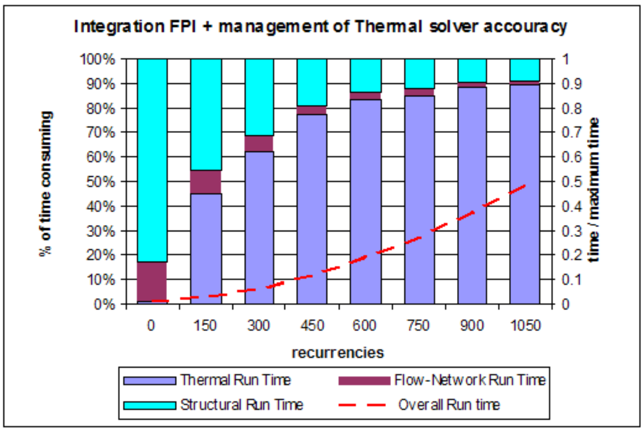


Figure 9 Comparison between percentage of required time spending for the integrated codes for simple FPI procedure and FPI + thermal solver accuracy management.

ENGINE TEST CASES

3) <u>Evaluation of the Impact on the design phase</u>

The first application of the integrated procedure presented, has been performed in order to evaluate the effects in terms of results and time consumption, even on a very easy test case.

In other words, we would answer the question riaised by the designer of the SAS, that is: " Which is the inaccuracy that we have without applying the integrated procedure?". Moreover, the test case also offers other information about how much the procedure costs, that in a Just-In-Time-Market, is equivalent to saying how long does the procedure take when it is applied.

For this reason, the component chosen for the simulation should be an easy, but also a significant application, like the static parts of a Secondary Air System, (see models in Figure 10).

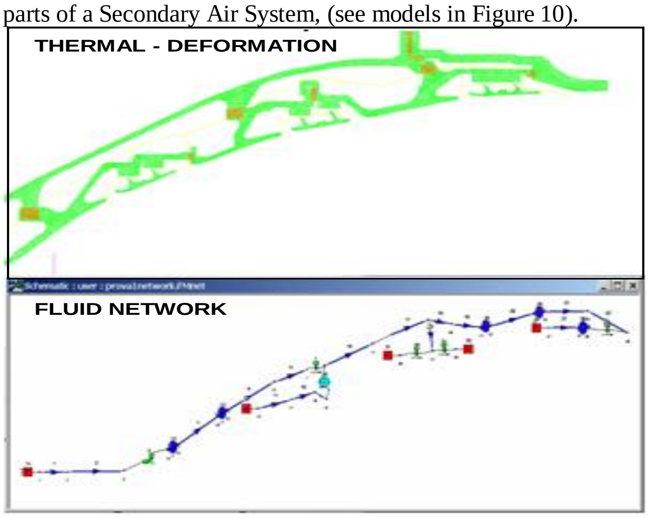


Figure 10 Sub-models used in the integrated approach

The dimensions of the models generated for simulating the component are hereafter summarized:

Analysis	Number of Nodes	Number of element
Structural FE	10000	15000
Thermal FE	10000	15000
Flow Network	40	30
Overall Coupled variables	All FE nodes temperature 10 chambers temperatures 10 pressure values 20 component mass flow values 10 geometrical deformation models	

Using the presented models, three different levels of analysis have been performed:

First test) only the thermal behaviour of the casing has been simulated (no FLUITHEST integration enhancement)

Second test) only the coupling of the thermal model and the flow network has been introduced.

Third test) Complete FLUITHEST integration

In the following pictures the contour plot of the differential temperature calculated for test 1 and test 2 (Figure 11) and test 1 and test 3 are presented (Figure 12).

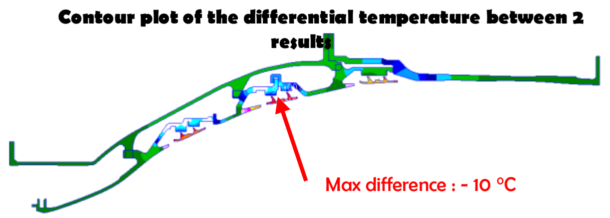


Figure 11 Differential results between first and second test

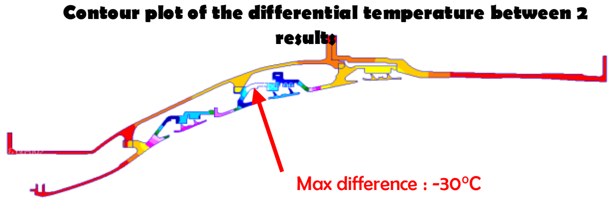
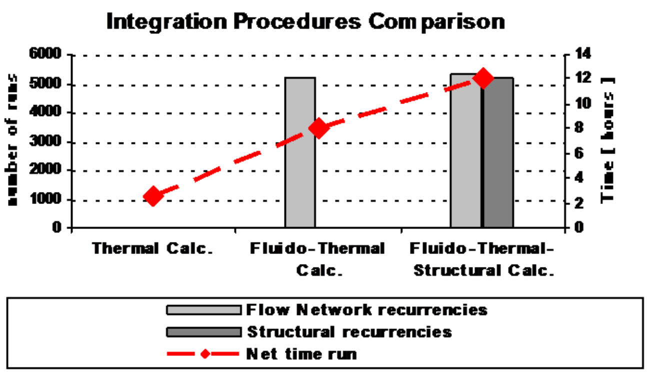


Figure 12 Differential results between first and third test

These analyses show that the integrated procedure allows the achievement of a progressive reduction of inaccuracy that is a consequence of the non hierarchical process.

Moreover, this increase in accuracy can be obtained only with an increase in computational time: the three test cases have been compared as in the graph below.



4) Prediction capability vs. experimental data

The prediction capability of the complete integrated code (FLUITHEST) has been verified by reproducing the thermal and pressure scenarios of an LP turbine rotor module for which experimental data were available in AVIO in the position marked in Figure 13

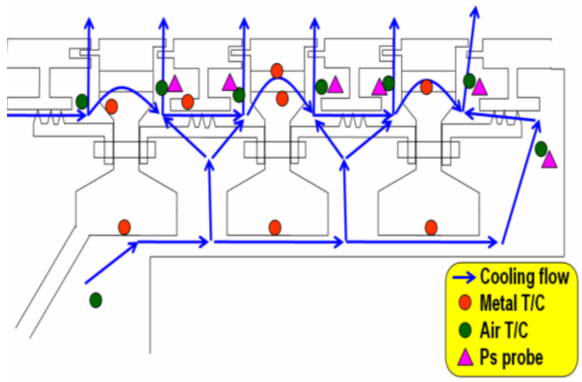


Figure 13: scheme of engine geometry and measurements points

FLUITHEST MODEL

Main characteristics are:

- Thermal Model: 9462 QUAD4 elements, 2312 HEXA8 elements, 11774 mesh nodes, flow network with 293 advection bars and 280 air nodes;

-Fluid 1D Model (see Figure 14): 373 TNODES, 212 orifice, 186 vortex, 4 seals, 48 cavities and 12 internal ducts.

- Structural MARC model: 5615 QUAD4 Elements, 7349 nodes, 240 MPC elements, blade and bolt centrifugal force simulation.

Reference module dimension (mean radius): 0.15 [m]

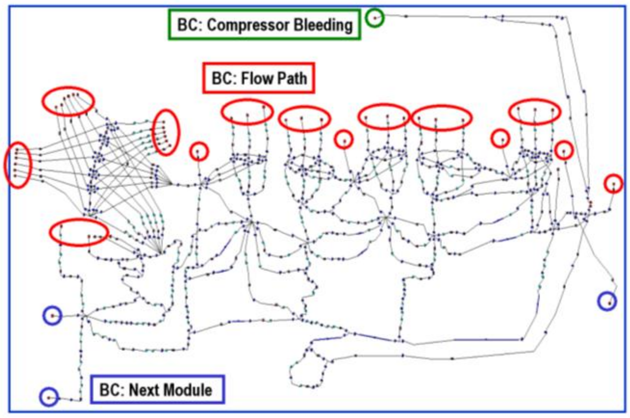


Figure 14*: Fluid 1D model*

Both Steady State and Transient [slam accel + slam decel] experimental data are available and both the conditions have been simulated using FLUITHEST approach.

STEADY STATE ANALYSIS:

Figure 15 and Figure 16 represent the differences between the experimental data (mathematically averaged in each sections) and the model simulation results for metal and pressure sensors applied on the module:

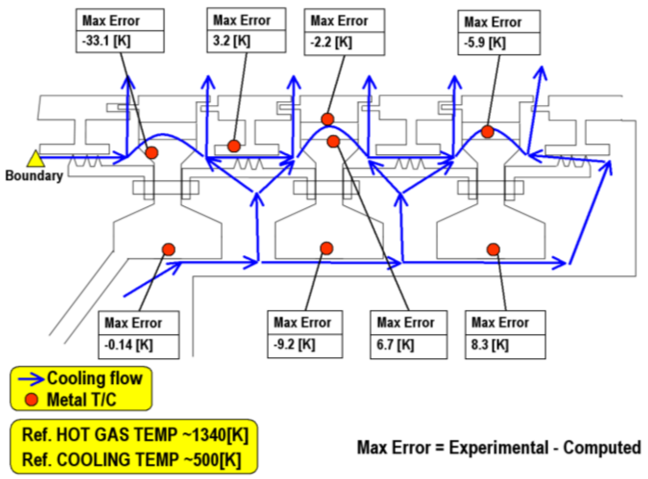


Figure 15*: Metal and air temperature scorecard*

A statistical analysis of these data shows:
A maximum difference of 33.1 [K] with:
An average error (all the points) of -4 [K]
A Std. Deviation of the differences (all the points) of 13.16 [K]

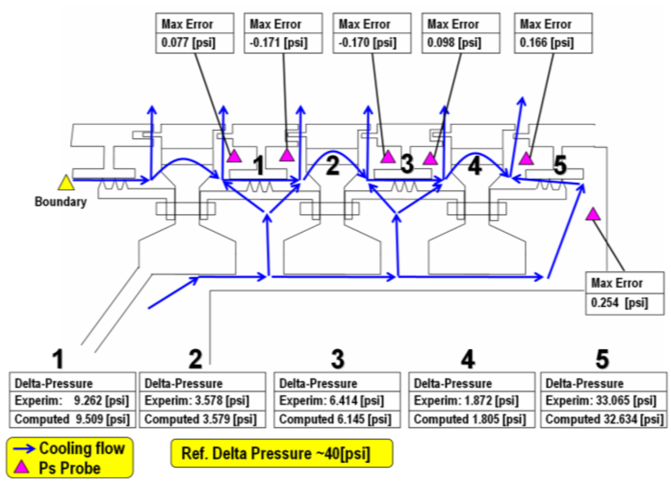


Figure 16: Pressure scorecard

The following graphics (*Figure 17* and *Figure 18*) show the statistical distribution of the differential results for metal temperatures and for all the measured temperature data.

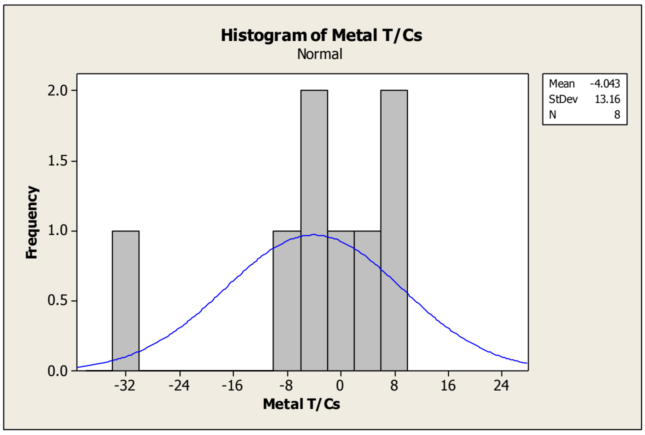


Figure 17: Statistic distribution of the differential results for metal temperatures

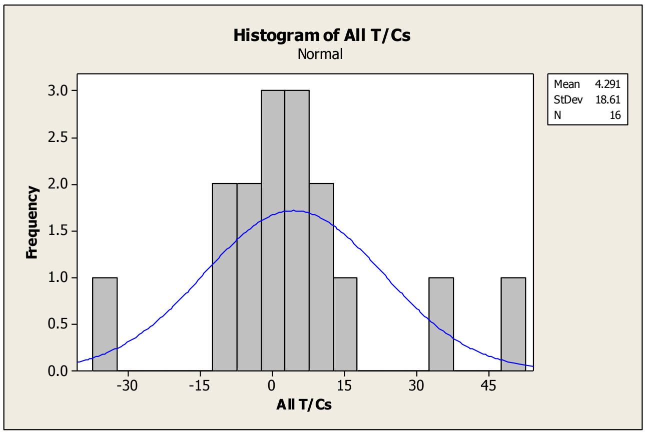


Figure 18 Statistic distribution of the differential results for all the measured temperature data

TRANSIENT ANALYSIS

A similar comparison has been performed in a transient condition to verify the capability of the FluiTheSt code to follow the behaviour of the pressure and temperatures in both slam accel and decel phases (vs. time).

The following charts are reporting non-dimensional values defined as:

- Temperature:

$$\frac{(ActualTEMP - AmbientTEMP)}{(MaxTEMP - AmbientTEMP)}$$

where:

- *MaxTEMP* is the maximum temperature value recorded for the specific sensor;
- *AmbientTEMP* = 70 [degF].

- Pressure:

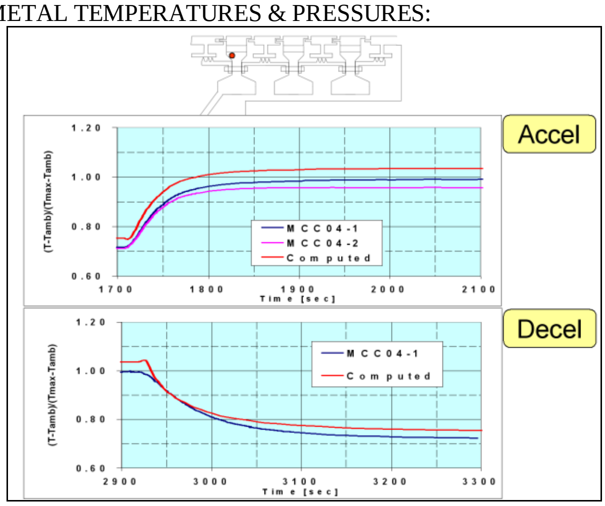
$$\frac{(ActualPRESS - AmbientPRESS)}{(MaxPRESS - AmbientPRESS)}$$

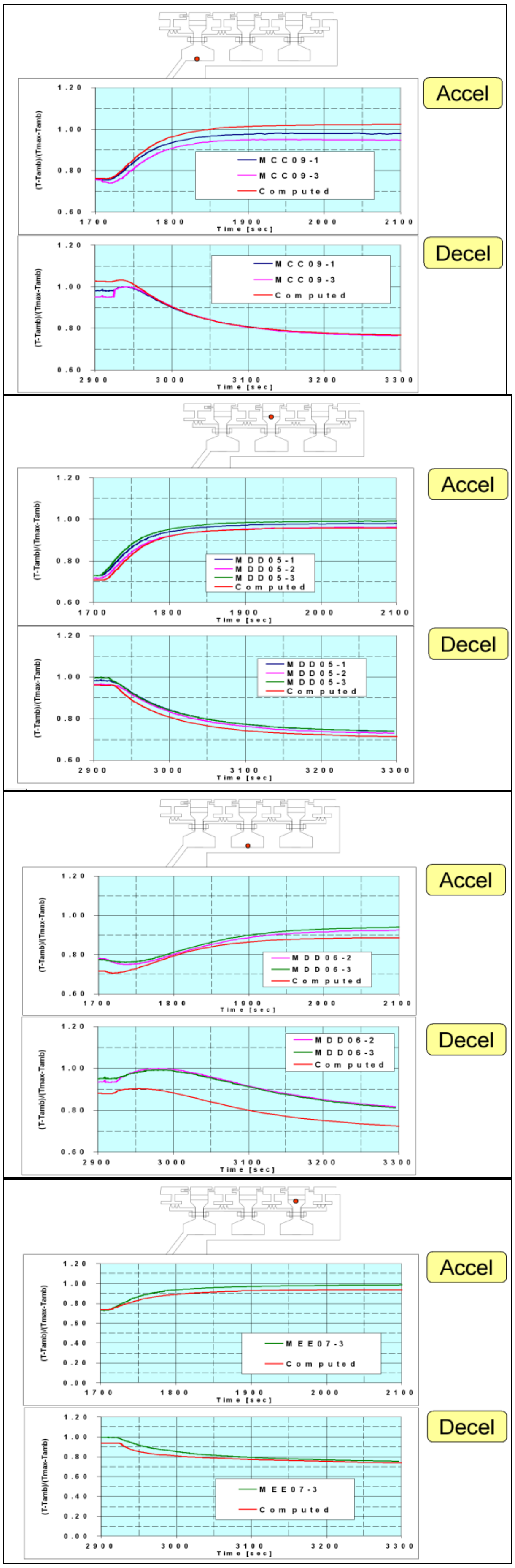
where:

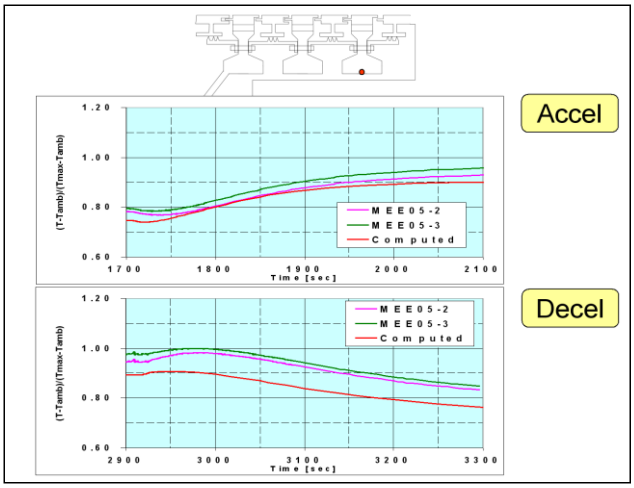
- *MaxPRESS* is the maximum pressure value recorded for the specific sensor;
- *AmbientPRESS* = 14.7 [psi].

Values of these parameters for each specific locations are hereafter reported.

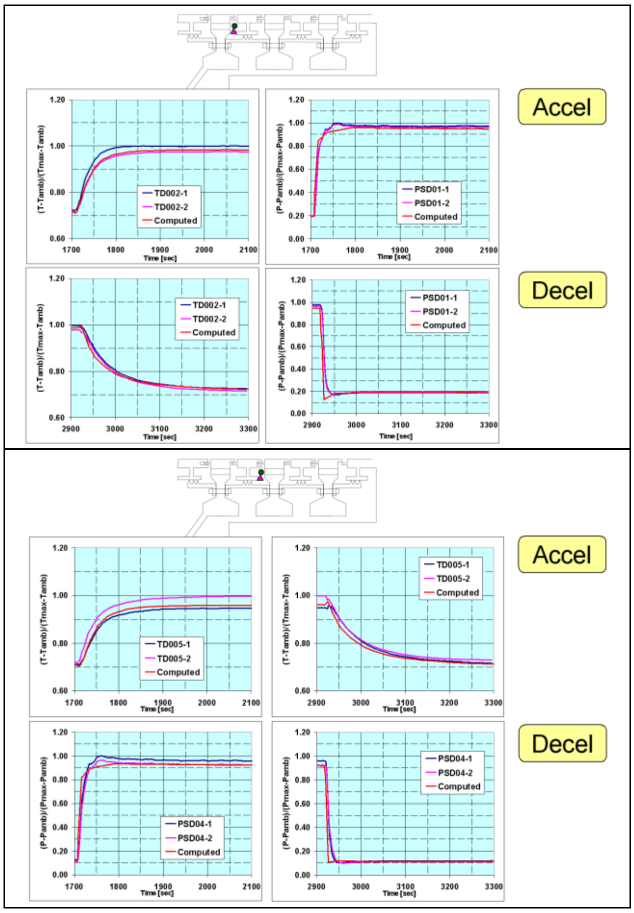
METAL TEMPERATURES & PRESSURES:

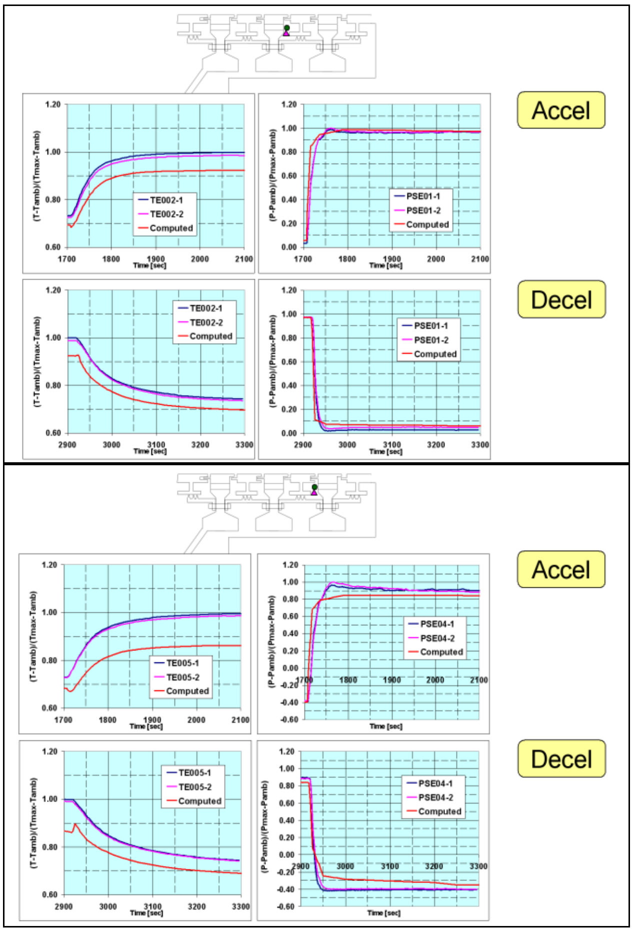






AIR TEMPERATURES & PRESSURES:





CONCLUSION

An innovative procedure for a multidisciplinary approach to a thermal design of a Low Pressure Turbine has been developed together with a dedicated tool.

Specific activities have been carried out in order:

- to check the code capability to represent the real phenomena with the desired accuracy
- to verify the impact on the design procedure both from an accuracy point of view and for computational time consumption.

Even if this last one is obviously increasing with respect to a single loop design, the next generation of aero-engine turbines will not be competitive without considering this kind of approaches and these approaches will not be implemented without automated and validated tools.

The next development phases of these methodologies will be to improve further the algorithms for reducing the running time, and will include the evaluation of more than 1D characteristics of the cavities fluid-dynamics.

REFERENCES

[ref. 1] Scott B. Lattime: "Turbine Engine Clearance Control, Systems: Current Practices and Future Directions", NASA / TM-2002-211794

[ref. 2] A.A.V.V: "Current state-of-the-art in Multidisciplinary Design Optimization", AIAA MDO Technical Committee, 1991.

[ref. 3] Balling, R.J.: "Sobieszczanski-Sobieski, J. 1994: Optimization of coupled systems: A critical overview of approaches", AIAA Paper 94-4339

[ref. 4] Hulme, K.F., and Bloebaum, C.L.: "Development of CASCADE – A Multidisciplinary Design Test Simulator." Sixth AIAA/NASA/ISSMO Symposium on Multidisciplinary Analysis and Optimization, Bellevue, WA, September, 1996, pp. 438-447.

[ref. 5] Cramer, E.J. et al. 1992: "An alternative problem formulations for multidisciplinary design optimization", 4-th AIAA/NASA/ISSMO Symp. on Multidisciplinary Analysis and Optimization (held in Cleveland, OH)

[ref. 6] K. English, C. L. Bloebaum, and E. Miller: "Development of multiple cycle coupling suspension in the the optimization of complex systems. Structural and Multidisciplinary Optimization", 22:268–283, 2001.

[ref. 7] Miller, E. 1995: "Coupling suspension and elimination in multidisciplinary design optimization. M.S. Thesis", State University of New York at Buffalo

[ref. 8] Haftka, R.T. 1985: "Simultaneous analysis and design", AIAA J. 23, 1099-1103

[ref. 9] Cramer, E.J. et al. 1994: "Problem formulation for multidisciplinary optimization. SIAM J. Optim". 4, 754-776

[ref. 10] Sobieszczanski-Sobieski, J., Agte, J., Sandusky, R. Jr.,: "Bi-level integrated system synthesis (BLISS)," AIAA-1998-4916, AIAA/USAF/NASA/ISSMO Symposium on Multidisciplinary Analysis and Optimization, 7th, St. Louis, MO, Sept. 2-4, 1998, Collection of Technical Papers.

[ref. 11] AAVV.: "MSC P-THERMAL User Manual", version 2004.

[ref. 12]AAVV.: "FLOWMASTER User's manual", version 6.5.

[ref. 13] Hulme, K.F., and Bloebaum, C.L.: "Development of a Multidisciplinary Design Optimization Test Simulator.", Structural Optimization, Volume 14, Number 2-3, October, 1997, pp. 129-137.