Proceedings of ASME Turbo Expo 2020
Turbomachinery Technical Conference and Exposition
GT2020
September 21-25, 2020, Virtual, Online

GT2020-16003

DEVELOPMENT OF AN INTEGRATED PROCEDURE FOR COMBUSTOR AERO-THERMAL PRELIMINARY DESIGN

Carlo Alberto Elmi[1], Fabio Agnolio[2], Roberto Ferraro[2], Anne Menard[3], Hauke Reese[4], Antonio Andreini[1]

[1]DIEF – Department of Industrial Engineering, University of Florence, Florence (FI), Italy
[2]GE Avio s.r.l., A.T.O. Combustor Design & Technology, Rivalta di Torino (TO), Italy
[3]ANSYS France, EMEA Field Engineering and Services, Montigny le Bretonneux, France
[4]ANSYS Germany, EMEA Field Engineering and Services, Darmstadt, Germany

ABSTRACT

In the development of an aero-engine combustor, the definition of a preliminary design is a practice in which know-how, product experience and design rules are focal in deriving a configuration able to meet the functional requirements. Several configurations, and hence multiple geometries resulting in different behaviours, are iteratively analysed in this phase to extensively explore the design space. In this context, an automated procedure ranging from preliminary design to life estimation is necessary and crucial. A framework in which the tools employed in the design workflow are integrated and the low-added-value tasks are automated can allow the reduction of time per analysis within the loop and the enhancement of the procedure's robustness.

In this paper will be presented the Combustor Design System Integration (DSI), a methodology aimed at easing and streamlining the design process of aero-engine combustors. To do this, digitization has been taken as the common thread for developing a data-centric approach. The logic behind the procedure will be reported, to focus then on the aero-thermal preliminary design. The procedure, for this phase, is composed of three main integrated components: a CAD generation system, which collects all the geometries for creating an exportable 3D model, a 1D thermal solver for the positioning and sizing of the aero feature on liners (i.e. cooling, dilution…) and a CFD environment with automated pre/post processing operations for reacting-flow analysis.

The aim of this work is to contextualize the DSI approach in the combustor design process and to provide a first description of the methodology designed and developed in GE Avio. For that purpose, a straight-through configuration -the lean combustor NEWAC developed in the homonymous EU project- will be exploited as a test case. The development of the procedure is still in progress, so a validation through test cell data comparison, as well as highly-resolved CFD results, will be the subject for future papers.

Keywords: Aeronautics, Engine, Combustor, Design Automation, Methodology, Digital

NOMENCLATURE

ACT	Application Customization Toolkit
BC	Boundary Condition
BoI	Body of Influence
CAD	Computer Aided Design
CFD	Computational Fluid Dynamics
CPU	Central Processing Unit
DSI	Design System Integration
FEA	Finite Element Analysis
FGM	Flamelet Generated Manifold
GE	General Electric
GUI	Guided User Interface
ICAO	International Civil Aviation Organization
MBPM	Metric-Based Process Mapping
NEWAC	NEW Aero engine Core concept
NOx	Nitrogen Oxides
NS	Named Selection
PDF	Probability Density Function
PERM	Partially Evaporating and Rapid Mixing
PrF	Profile Factor
PtF	Pattern Factor
RANS	Reynolds Averaged Navier Stokes
RSM	Remote Solver Manager
TBC	Thermal Barrier Coating
XML	eXtensible Markup Language
UDF	User Defined Function

INTRODUCTION

From an industrial point of view, time-to-market is a crucial aspect which is becoming increasingly important in many fields, including aviation. If high requirements act as a catalyst for the technological maturation, the demand for making advanced solutions available at product level can shorten the development phase. This leads all the actors belonging to an aircraft program -from designers to manufacturers, from airplane structure to engines- to optimize their processes. In such a challenging environment, introducing methodologies and tools devised to save time and resources is strategical.

For this purpose, several procedures have been developed for the sole combustion system. *Honeywell Engines & Systems* started implementing the Advanced Combustion Tool in 1998, an integrated process for rapidly analyzing and optimizing the combustor's aerodynamics from compressor deswirl to turbine stator inlet [1]. *Rolls-Royce* too supported the development of such methodologies. In PRECODES (Preliminary Combustor Design System), design rules and parameters were formalized in a knowledge-based system for determining the flame tube concept and then feeding the preliminary phase [2–5]. A more detailed approach is represented by PROMETHEUS Design System [6,7], in which pre-and-post processing operation are automatically set according to a feature recognition method. For all of these works, as well as other examples coming from the Academic field [8–10], it is clear that procedures and then tools are adopted to evaluate the aero-thermal behavior. This is the reason why an integration between CAD and CFD processes results is central: by relying on early-stage design tools (from correlative, 0-1D approaches) a 3D geometry can be automatically generated and prepared for the fluid dynamic analysis; boundary condition regions are then applied on the resulting fluid domain, which is meshed and sent to CFD simulation process.

Methodologies such as those mentioned above not only represent the state-of-the-art of integrated-and-automated procedures, but also the minimal standard for the industrial environment in which combustors are designed, developed and manufactured, such as at *GE Avio* company. It is therefore reasonable that such a beneficial approach should not be confined only to the aero-thermal evaluation, and indeed extending such philosophy downstream in the design process can significantly increase the overall value [11]. The Combustor Design System Integration (DSI) was created precisely with this intention: to rely on a data-centric approach in which information about the configuration, with special regard to geometry, is coherently collected and made available to the whole design flow. As it will be seen, the CAD model assumes a predominant role within the proposed methodology. The generation of the 3D geometry is an activity in which the designer is called to translate experience and best practices in a feasible solution. The arrangement of the cooling system represented a critical process. The hole feature must not only undergo specific constraints but can also have optimal dispositions which maximizes the cooling effect. With the aim of solving this step, a first effort has been made in rationalizing such best practices. After that, these rules have been implemented into the CAD tool in order to generate an appropriate model for CFD evaluation.

This paper will focus on the development of the aero-thermal methodology for preliminary design, the heart of the DSI procedure and so a leading topic in the implementation steps. To identify the purposes and basic concepts of this framework, the first section is dedicated to the contextualization in the combustor design workflow of the Tools from which it is composed. A demonstration of the procedure's functioning is then reported in the second part by means of *GE Avio*'s NEWAC test case. For this concept -a straight-through, lean combustor developed in the context of the homonymous European research project- the steps and results of processing it within the DSI are reported. Given the descriptive nature of this work, a validation of results based on test-cell data comparison and hi-fidelity simulation is not reported, considering this topic to be a matter for a future, dedicated paper.

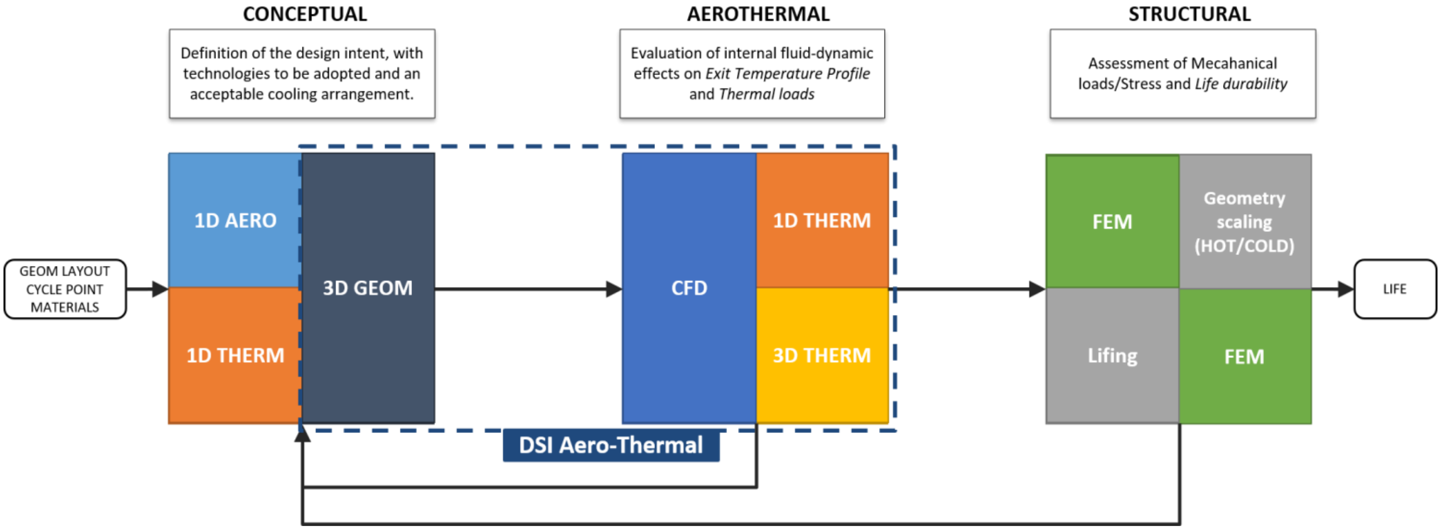


FIGURE 1: A SIMPLIFIED REPRESENTATION OF THE STEPS INVOLVED IN THE COMBUSTOR DESIGN

1 Combustor DSI Methodology

As is well known, the combustor is a critical component for the whole engine. It is by means of the reaction process that the energy contained in the fuel is converted into enthalpy for the working fluid, which can then be extracted to realize the propulsive effect. The combustion, however, is not so easily realized, nor even controlled. In fact, in maximizing the combustion process for performance, the resulting stresses may be challenging for component durability.

With reference to Figure 1, a summarized description of the different phases involved in the design process can be given.

The very first step is devoted to determining a configuration which can meet the product's basic requirements. A layout suitable for the specific application, as well as a down-selection of the technologies to be employed, is defined here by means of design rules and correlative tools. This 0-1D level information represents the standard input for generating a 3D preliminary CAD, on which a first feasibility check can be performed. Geometrical guidelines, derived from structural or manufacturability evaluations, are at this level adopted to find an ideal arrangement for specific features (i.e. cooling or dilution holes). This resulting configuration becomes the reference model, on which more detailed analysis can be performed.

The aero-thermal evaluation represents the second phase of development, in which predictions of flow/flame behavior and hardware temperatures are carried out. To do this, a CFD analysis is exploited to simulate internal aerodynamics and reaction processes. Its results provide both an estimation of aero functional key parameters (gas temperature and its distribution at exit port, emission levels) and boundary conditions for the thermal assessment. A Finite Element Analysis (FEA) is then performed to calculate the hardware's temperature level considering heat loads of different nature.

Lastly, the set of loads coming from different disciplines - aerodynamics, thermal and mechanical- are considered together to verify stress limits with a structural FEA. A cyclic application of these loads, simulating the transient behavior during its mission, provides a life estimation.

This description highlights the underlying complexity in developing this product, which is configured not only in its multidisciplinary nature but also in the conflicts that can occur between the multiple requirements. If the first issue mostly deals with the communication between several tools involved in the design flow, the second one implies the iterative nature of the process itself. In such an intricate procedure, we can understand how digitalization may increase the overall value of the process -and hence- the product [12]. In this context, it is not only important how quickly results are generated but also the reliability of the information that is processed and, not least, the possibility of making such information available globally within process. The Combustor DSI procedure has been developed to satisfy these requirements, streamlining the design process while enhancing its robustness. In the following, the application of this philosophy to the aero-thermal methodology, with a focus on the tools adopted -and developed- for the preliminary design phase, will be described.

1.1 DSI for the Aero-Thermal Design

From an engineering standpoint, the preliminary design represents the phase from which a first overall system configuration can be defined. In this phase are included all the analytical evaluations which provide a feedback on requirements not immediately assessed into product's conceptualization [13]. This aspect often results in switching from correlative approaches, embedded in 0-1D design tools, to computational and more detailed evaluation that takes the discussion to the 3D level.

As concerns the combustor's aero-thermal design, it means that CFD and Thermal FE analyses are necessary to predict parameters such as:

- **Gas temperature**, explicative for requirements as Profile Factor (PrF) and Pattern Factor (PtF);
- **Emission levels** in terms of Carbon Monoxide (CO), Nitrogen Oxides (NOx), Unburned Hydrocarbons (UHCs) and soot production;
- **Material thermal conditions**, as the maximum temperatures resulting on combustor's components.

In a simulation-based design, the adoption of an iterative process is inevitable in order to fulfill all those requirements simultaneously [14]. This leads to multiple repetitive and low-value-added operations that must be performed within each design loop, tasks that involve user intervention and that can hence become sources of human error. With the objective of speeding up those activities and making the process more

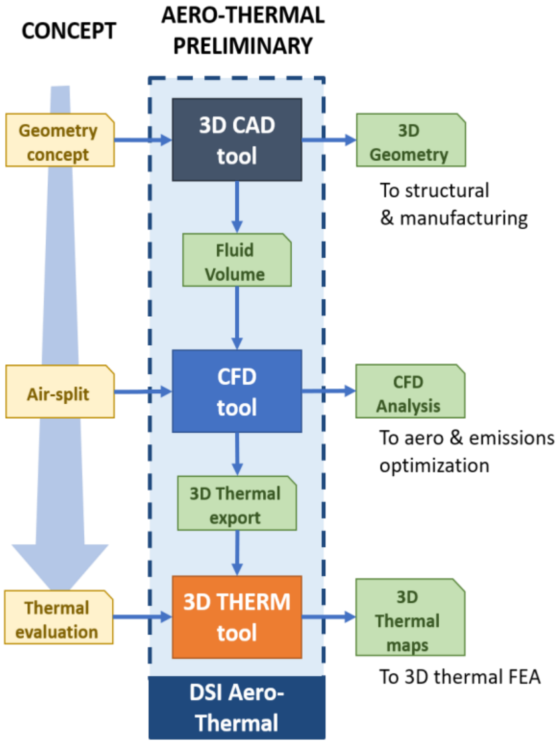


FIGURE 2: DETAIL OF THE COMBUSTOR DSI METHODOLOGY FOR AERO-THERMAL PRELIMINARY DESIGN

reliable, three different "tools" have been developed inside the DSI framework. All of these comprise scripts developed in the Application Customization Toolkit (ACT) provided by *ANSYS*, which allows the customization of off-the-shelf product capabilities to specific functional needs. As follows, these extensions rely on a standard input/output logic that effectively enables the linking between the different tools. Dedicated routines are also implemented for automating pre- and post-processing activities, as well as including best practices and design facilitations in the development phase.

1.2 Tools

In Figure 2 is depicted a schematic of the Combustor DSI methodology dedicated to the aero-thermal preliminary design, each tool in which is discussed in this subsection.

Section 1.2.1 is dedicated to the combustor's CAD model generation, which means the process of preparing the 3D model starting from preliminary design information, as well as the tool's functionalities that have been specifically developed to carry out this task.

Then, in Section 1.2.2, the workflow for preparing, executing and elaborating the CFD analysis of the combustor is detailed. A specific focus will be given on the structure of the tool, designed to collect and rationalize both the design information and the numerical models to be employed in the simulation.

In Section 1.2.3, lastly, the steps required for creating the set of thermal boundary conditions necessary to perform a 3D thermal FE analysis are reported. These are generated starting from the results of the 1D thermal tool employed for assessing the combustor metal temperature and designing the cooling system on liners.

To evaluate the amount of temporal and computational resources required by the design process, Section 1.2.4 describes the Metric-Based Process Mapping (MBPM) method. This strategy, which has been used for identifying and planning improvements within the process, is going to be adopted in this work as a metric for estimating the resource savings.

1.2.1 CAD generation

The goal of the customized CAD tool, whose general process is shown in Figure 3, is to generate a geometrical model which is representative of the combustor. Within the DSI, such 3D geometry accounts for the most significant element of the methodology: it represents the repository of all the information regarding the product's configuration, and hence is taken as reference for building all the geometrical input for the downstream process. The tool has been developed in *ANSYS* SpaceClaim, a fully scriptable program designed to create and adjust CAD geometries. To export the proper information for the different simulation disciplines involved in the design, a set of dedicated routines have been implemented. Up to now, the SpaceClaim script is able to export the parameters necessary to manufacture the cooling system, and a properly defined fluid domain on which a CFD analysis can be performed. Given the topic of this paper, this last process will be considered.

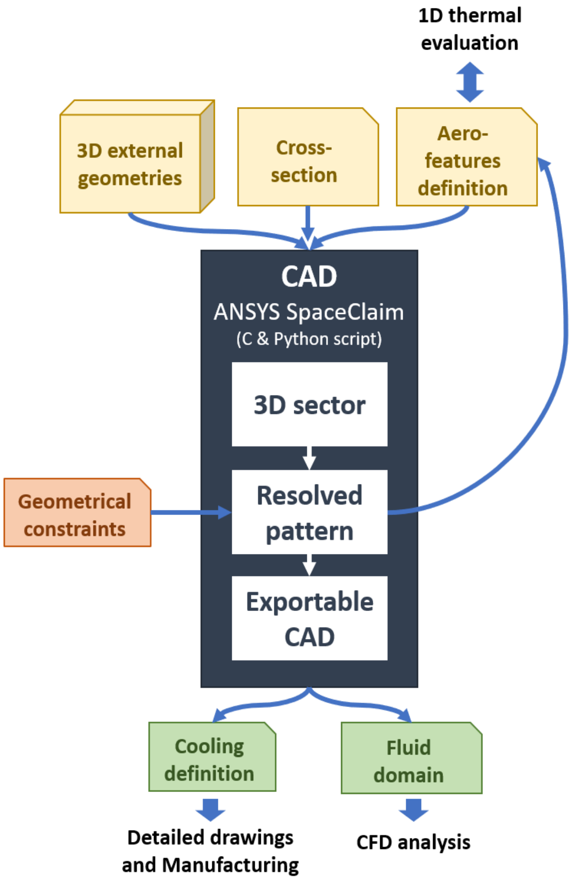


FIGURE 3: THE CAD PROCESS

The starting point for the geometry generation is the output of the 1D design tool, so-called Therm-1D, which allows definition of the most appropriate cooling system and arrangement to adopt, considering the combustor's cross-section definition and the air split for a certain design point; this will be described in depth in the next paragraphs. The main challenge for building the 3D model is to arrange the different hole features, as long as only a few physical constraints can be considered in such a simplified approach. The additional rules, which mostly address solving the collision problem between geometrical bodies, can be grouped as follows:

- Structural constrains
- Fluid dynamic constrains
- General design constrains

The first constraint ensures a minimum distance between neighboring holes and is critical, because in case of violation the design can break.

The second one includes the cooling concept itself. Such constraints are that holes in the jacket should not be placed on top of holes inside the liner, or that holes in the liner should be staggered properly for creating a continuously cooling film.

The last constraint regards the design process itself, i.e. moved, added or deleted holes that can affect the functioning of the

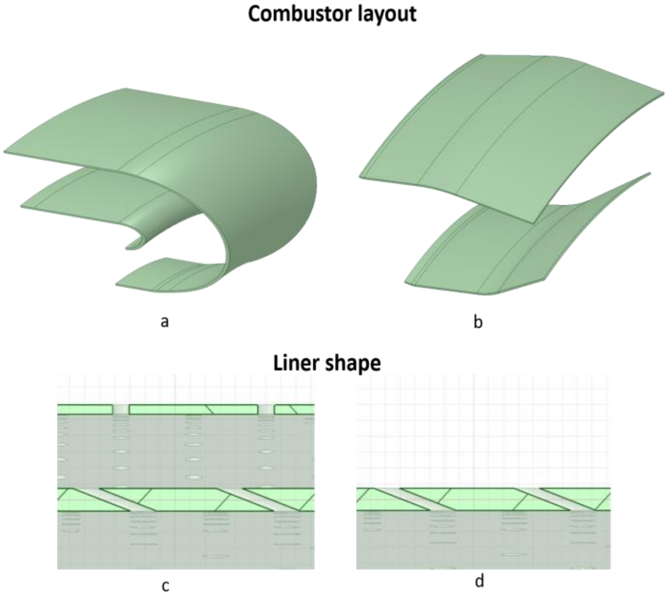


FIGURE 4: FEASIBLE GEOMETRIES BY THE CAD TOOL

combustor by modifying the pressure drop and the air-split between the different features.

Within the procedure, up to four different types of hole are available for the design:

- dilution holes, to feed the flame tube so the quenching effect can be realized;
- multi-holes, for effusion cooling effect;
- impingement holes, that realize a cooling effect by jets impinging on the cold side of a component, such as liners and/or heat shields;
- nugget holes, to produce a film coverage from a slot geometry.

To complete the model generation, and hence realize an exportable CAD, all the geometrical features that are not addressed by the 1D design tool can be imported from dedicated libraries. Swirlers and heat shields, as well as mounting flanges and casings, can be selected as 3D external parts according to the design purpose.

According to the CAD tool functioning, the first activity performed is a general check over the input coming from the conceptual design phase. This has the dual purpose of rationalizing the information required to define a repeatable combustor sector, representative of the full annular configuration, and analyzing the consistency of the cooling definition derived from Therm-1D. For the latter case, the number of holes constituting the rows is adjusted in order to obtain a symmetric and repeatable cooling pattern upon the complete configuration. The reference sector's extension is determined by the number of injectors in the configuration.

Once the input passes all the preliminary checks, the combustor shape can be generated. As reported in Figure 4, the flame tube layouts enabled by the automatic procedure are both reverse-flow (a) and straight-through (b); the liner shape can come as double (c) or single wall (d) on which thickness, as well as Thermal Barrier Coating (TBC) application, can be fully defined by the designer. The positioning of the different holes can be now addressed at a 3D level for each different liner location. A hierarchical approach between the different types is used to perform this second tool's main task: the dilution holes are placed first, then the effusion holes are arranged around these and, lastly, the impingement holes are placed considering both the previous arrangements. Different strategies have been developed to automatically solve the problem of intersection between these geometrical features, with the purpose of providing the best cooling distribution in regions with limited space. Dealing with minimum distance requirements and optimal arrangement for cooling effects, these rules are based on the relative position between the effusion hole rows and dilution holes. According to different situations, each cooling hole can have an axial or tangential sliding, but even a change in the impact angle can take place. In case of a double wall liner, the collision problem (and hence strategies for solutions) is extended to impingement holes too, that come up in conflict with the effusion cooling arrangement. Nevertheless, the problem can even get more complicated in regions of high curvature of the liners. A strategy to solve all the collisions in a single step has proved to be too complex, so an iterative process has been implemented. During the loops, the distances between neighboring rows is estimated. In case of minimum distance being reached, the holes are moved forward or backward until the minimum distance criterion is satisfied. According to test cases of interest, most of the pattern results as resolved with about three iterations. The designer can also take control of these movements with a manual procedure, since an interactive interface has been implemented as well. All those changes are driven by specific commands recorded in dedicated input files, which represent the starting point for a more complex optimization strategy that is currently under development. Once the cooling holes are all set, a post analyzing routine is used before the last major activity: the exporting. Each row position of the resolved pattern is averaged according to the coordinates of the holes that belonging to it. The values are compared once again with the initial input values: if there is a gap that exceeds the accepted range, an update of the 1D design tool needs to be performed for consistency inside the design loop.

The export for the CFD analysis represents the final task of the automated CAD process. Via a dedicated routine, a ready-to-mesh fluid domain is created according to the design specifications that are needed. The volume is generated by intersecting the exportable CAD model, as described above, with a standard starting domain on which "static" named selections are defined. The set of "dynamic" named selections is instead retrieved by the specific design, i.e. cooling hole inlets and dedicated planes for post-processing. Since features such as injectors and dilution holes are explicitly resolved, this phase defines and places the proper feeding plenums to be merged to the main domain. The rest of the information required by the downstream process is collected into a python library file.

1.2.2 CFD analysis

The goal of the customized CFD Tool is to enhance, streamline and automate the whole CFD workflow. The automation of pre/run/post-processing operations enables the reduction of the human and CPU time per analysis loop, the enhancement of the procedure robustness and so its optimization.

The CFD system automation is done within the *ANSYS* Workbench environment using *ANSYS* ACT, a fast and easy way to tailor simulation workflows based on easy-to-learn yet powerful XML and IronPython programming languages.

In the DSI framework, an ACT application has been developed to automate and customize the full routine CFD workflow and to integrate data with the *ANSYS* product line. This application comprises four wizards with different purposes. With reference to Figure 5, a summary description of the wizard scopes (yellow boxes) involved in the CFD process can be given.

A first wizard has been developed to collect the data coming from the upstream processes, to organize and prepare them for the complete workflow. The CFD input data are synthetized in a file: boundary conditions and their characteristics (mass flow, inlet angle, temperature), injection properties, monitor definitions and data for the post-processing.

The second wizard has been developed to cover the whole CFD process. It allows users to both automate and customize the whole stream and it is made of the steps reported below:

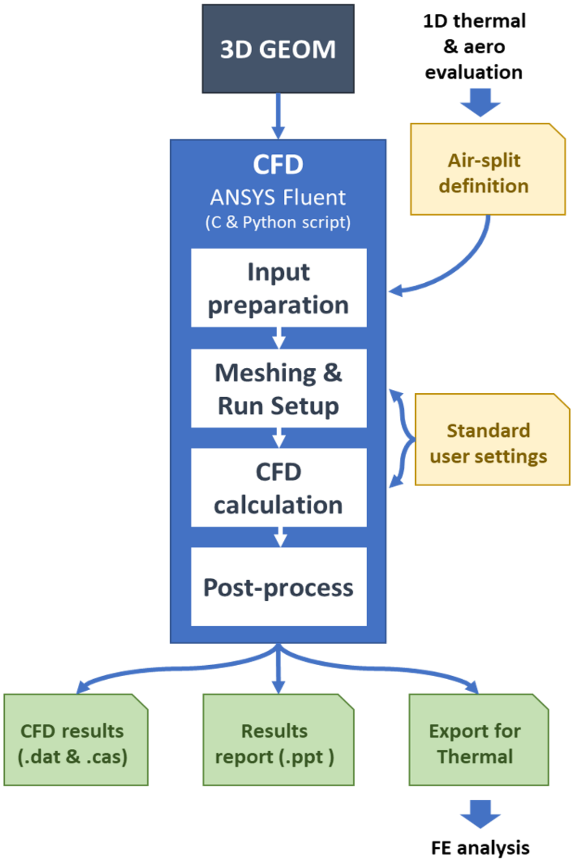


FIGURE 5: THE CFD PROCESS

- CFD Mesh
- CFD Setup
- CFD Calculation
- CFD Post-processing
- CFD Report.

This wizard walks non-expert end users step-by-step through the simulation, each of them asking the user to define the required information.

For the CFD Mesh step, the Wizard Guided User Interface (GUI) collects all the main mesh characteristics and the number of cores for parallel meshing the fluid domain generated by the CAD tool. The wizard also offers the flexibility to create tetrahedral, polyhedral or poly-hexacore meshes. The latter two, generated in *ANSYS* Fluent Meshing, provide a comparable accuracy in terms of results whilst minimizing mesh size and maximizing cell quality.

The CFD Setup requires some generic *ANSYS* Fluent files (UDF, scheme, pdf table, interpolation files). In this step, if an interpolation file has already been generated it can be used as an initial solution, otherwise a hybrid initialization method is set as default. To save CPU time, a convergence criterion based on outlet temperature monitors can be defined.

The iterations number and the *ANSYS* RSM (Remote solver Manager) submission parameters are required to define the CFD Calculation step. RSM enables the automatic submission of the calculation job to a remote cluster for the parallel computing.

In the CFD post-processing step, a list of post-processing outputs is submitted, such as contours, profiles, probes, tables, etc... This step manages the data export creation required for the downstream processes.

Once all the wizard fields have been filled, the python code performs the complete CFD workflow in a fully automated way without any human intervention: from the creation of the mesh to the filling of a PowerPoint report in which all the exports of interest are summarized so it can be used for checking activities.

The wizard offers the flexibility to exit the full automated process at several steps: after the setup definition, after the run calculation or after the post-processing results. Then a third and a fourth wizard are available to cover different scenarios and continue the workflow from the current state to automate the missing steps. One is dedicated to the automatic creation of the post-processing and the PowerPoint report; another one is dedicated to the automatic creation of the PowerPoint report only. Thanks to those two wizards, users can easily bypass the default RSM servers to run on other machines, run further iterations or create additional specific post-processing to be added in the report.

1.2.3 FE Thermal evaluation

Automation in generating the thermal boundary conditions for a thermal FEA, the process for which is depicted in Figure 6, is achieved using two steps that involve different tools.

The first step involves a FORTRAN script for elaborating input and output of the thermal conceptual design tool, Therm-1D [15]. This is an in-house routine developed by University of Florence, a 1D conjugate heat-transfer solver which relies on a

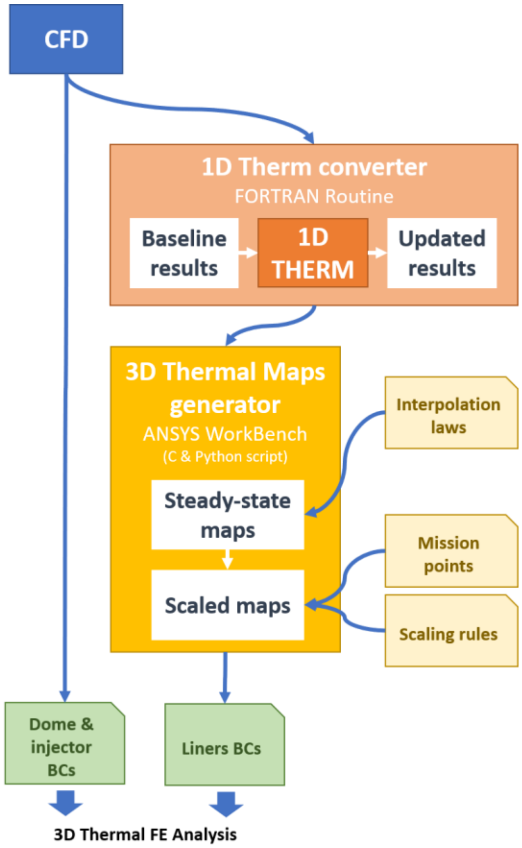


FIGURE 6: THE FE THERMAL PROCESS

correlative approach to exploit the calculation on liners. Since both the fluid and the solid are considered, this tool is suitable for defining cooling and quenching systems. Together with the 1D hole pattern, related results in terms of metal and coolant temperature, as well as of heat transfer coefficients, are given. The pre- and post-processing script not only allows the linking with the CAD and the CFD tools, by providing the definition of the holes and the coolant temperature respectively, but also with the 3D thermal evaluation. In fact, specific thermal quantities (wall metal temperature, gas radiative temperature, heat transfer coefficient…) are extracted and prepared for the different angular sections in which the Therm-1D networks have been created.

This represents the input to the second tool, an ACT extension developed in *ANSYS* Workbench environment. The goal of this procedure is to create 3D cloud-points of thermal boundary conditions that can be mapped in a thermal FE model. To do this, the wizard comprises three main steps that can be managed by user interface or in batch mode, to fully automate the following operations. A first, preliminary activity is dedicated to specifying the resolution of the map, as the number of points in the axial and tangential direction, while linking the Therm-1D results to be adopted. The generation phase can so be managed, in which rules to be adopted for interpolating 1D results at the 3D level must be defined for the different locations (outer or inner liner). The result is a set of 3D distributions of quantities of interest, in the specific mission conditions of the 1D model. Furthermore, in the last steps, such maps can be scaled on different mission points by means of correlation and best practices. It provides the input for the transient thermal assessment, giving the desired flight mission. An additional plug-in enables visualization of the maps, which is useful for checking and comparison activities.

1.2.4 Resource savings estimation

To evaluate the temporal and computational resources saved by exploiting the Combustor DSI approach, a Metric-Based Process Mapping [16] has been adopted. The MBPM is a practice developed in the context of lean management that is typically used for analyzing and documenting an existing process, in order to identify areas for improvement and to strategically define actions for its enhancement. Together with mapping the value across a process, this specific method includes information about key times and quality metrics. Such details are necessary to drive changes, therefore MBPM has been chosen instead of the Value-Stream Mapping method [17], despite the latter being more streamlined and therefore easier to implement for the present case.

The first step for implementing MBPM for the combustor design process has been to map the “current state”. This is representative of the “standard” approach adopted in *GE Avio* for carrying out the preliminary design activities, ranging from receiving conceptual design input to delivering thermal boundary conditions for FEA analysis. Two different schemes have been created:

- “new design” activity, in which phases of building and debugging of models for the different analysis are included,
- “re-design/re-run” activity, intended for design loops in which major/minor modification can be addressed for an already existing combustor model.

The second step consists of depicting the “future state”, which is the result of designing and implementing all the improvements offered by the Combustor DSI procedure.

The results expressing savings between the “standard” and DSI approaches have been estimated by relying on a direct comparison between the “current” and the “future” states, considering both the schemes identified for the preliminary design phase. Since maps are models of the process, a reliability-check over a real application has been performed first.

2 DSI Application: NEWAC Test-Case

In order to explain and demonstrate the functionalities of the presented procedure, in this section the results of applying the DSI procedure to a real-case configuration are reported: the NEWAC combustor, developed in the purpose of the “New Aero engine Core concept” EU project.

The prototype proposed by *GE Avio* is shown in Figure 7. The combustor, a lean-burn concept with a straight-through layout, makes use of the PERM injector (Partial Evaporation and Rapid Mixing) to drastically reduce the NOx formation. This advanced injection system is a co-rotating, double radial swirler

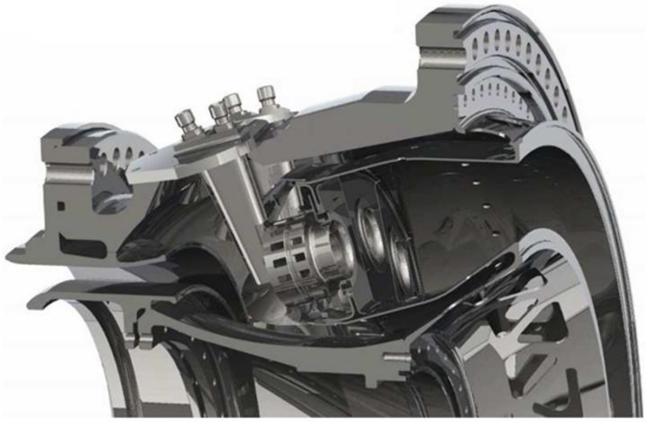


FIGURE 7: *GE AVIO*'S NEWAC COMBUSTOR

with a fuel staging between a pressure atomizer (pilot) and a film developed on inner's lip surface (main). This arrangement enables a correct stabilization of the flame during its functioning. The reaction process, and hence the NOx formation, is also modulated by the injection of air through dilution holes, realized on metal liners and placed in the final section of the combustion chamber. To protect the dome plate from high temperatures, a heat shield with a back-impingement cooling system has been provided. Purge air coming from this feature is adopted to produce a film covering effect in the very early section of the liners. A multi-hole system has been designed and displaced to keep such metal parts under acceptable thermal limits.

Since several studies, both in the numerical [15,18,19] and in the experimental field [20], have been performed on this specific concept; it lends itself well to demonstrating the procedure's functioning as well as being a benchmark for the next validation step and possible future developments. The outcomes reported in the following are representative of a "from scratch" preliminary design of the test-case, in which the framework and tools are used for generating a CAD model, performing a CFD analysis and preparing a set of boundary conditions for thermal FE analysis. The amount of resources saved are evaluated via an MBRM method.

2.1 CAD results

The process carried out to generate the 3D model can be considered to be simple, composed of three main steps.

The first step allows the combustor layout shown in Figure 8a to be built up. Input to this task is represented by the liner's shape, which is here given as a cross-section spline and external 3D parts. The latter comprise components that are not supposed to change during the preliminary design phase: for the specific case, injector and dome/heat shield. Liners are generated by revolution around the combustor's primary axis, considering the cross-section definition coupled with the thickness of the liner. The angular extension is calculated according to the number of injectors (i.e. 20° for 18 swirlers) so that a repeatable and representative sector of the combustor is created. Finally, the previously mentioned additional geometries are properly placed.

The second step consists of the topological optimization of the aero feature defined on the liners, which is the most critical process and the main goal of this CAD automated procedure. Information about the cooling and the quenching systems results from the preliminary thermal design are provided by the previously mentioned Therm-1D and stored in an excel repository file. Different holes are arranged considering feasibility requirements, following dedicated rules that are based on academic research and design lessons learned. The specific correction for the case of Figure 8b consists in an adjustment between dilution holes. Several and more complicated rules are implemented and can be chosen by the user. The result, which is obtained iteratively but automatically, is a "drilled" CAD model

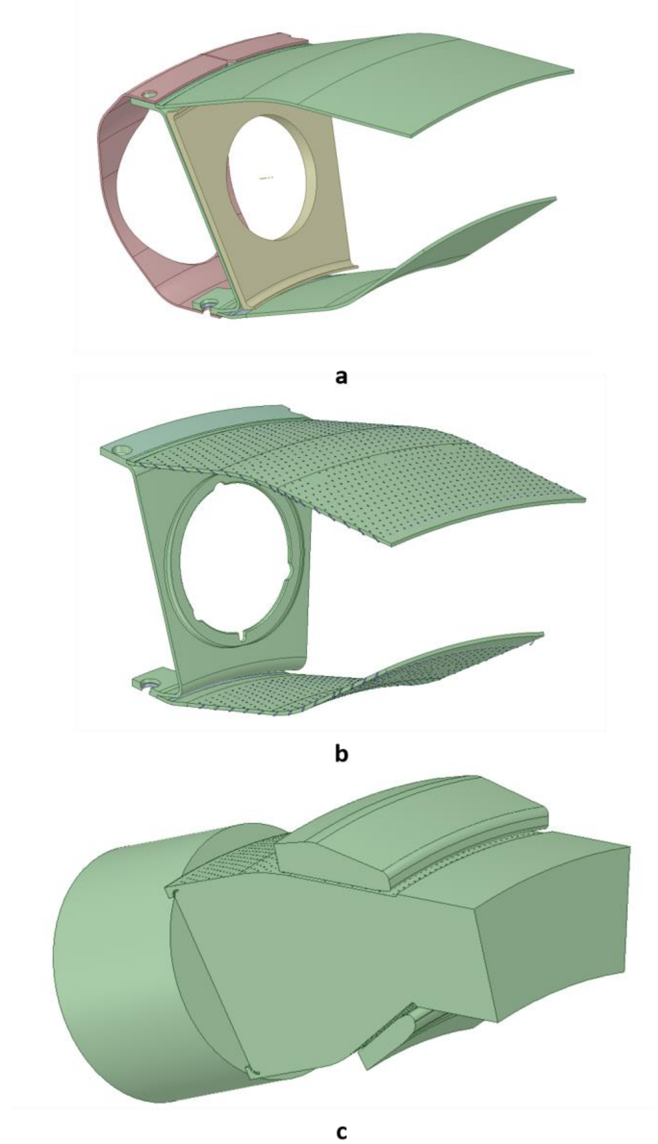


FIGURE 8: RESULTS OF GENERATING THE REPETITIVE SECTOR (a), DISPOSING DILUTION AND EFFUSION HOLES (b) AND ECPORTING THE CFD VOLUME (c)

in which interpenetration between hole bodies and minimum distances between hole objects are respected.

The final phase makes it possible to export such configuration for a CFD analysis. To this end, the geometry is properly defeatured and refined so mesh quality is ensured and only required features are considered. For the specific step, the resulting fluid volume is a simplified geometry in which cooling features are represented as footprints while injector and dilution holes are explicitly resolved. The feeding plenums are properly defined and placed on the model. Furthermore, bodies of influence (BoI) for mesh refinement, as well as named selections (NS) to set specific boundary conditions (BCs), are defined. The result is a fluid domain which complies with the standards required by the CFD model, including dedicated pre and post processing locations that can directly be retrieved from the geometry (Figure 8c).

2.2 CFD results

The preliminary activity for pursuing the CFD process consists in binding the CAD entity with the physical input from the conceptual design. Basically, the BC type, and if specified its value, is associated to the corresponding named selection present in the fluid domain. The constant linking with the 1D aero and thermal tools is realized by keeping the same naming, while the type is automatically recognized by means of specific NS classifications. For this case, as an example, a mass-flow inlet at the injector location and a pressure outlet at combustor exit is adopted, while all cooling holes and bleeding sections, which are represented as footprints on fluid volume surfaces, are treated with a mass-flow inlet/outlet respectively. There is also a set of additional information, at this level, that requires an input from the user. Those are the mission point, selected as Approach 272 ICAO test condition, and the injection system characterization, here specified with a double injection to best represent the behavior of the PERM system; also, most of the details needed for the post-processing phase are here requested.

All this input is stored into a recap excel file that can be specified once in the process (i.e. model preparation phase) and it is not supposed to change during the iterations. This file feeds the ACT of the pre-processing phase that manages two macro activities fully customizable from the tool's Graphic User Interface: the automatic meshing and the journal file creation. For the first one, a polyhedral mesh with a specific minimum element size and growth rate has been chosen, inflation is set on all wall boundaries and sizing for the BoI is extracted from the recap file. The resulting mesh is shown in Figure 9a. The second includes the general setup for the processing phase: the CFD numerical models implemented here will produce a steady state numerical result of the reaction field of the combustion chamber. Compressible Navier-Stokes equations are solved considering a reacting mixture made by air, Jet A-1 and combustion products whilst turbulence is modelled with the k-epsilon model. The FGM method is adopted to compute the reaction process so a PDF of the species is provided.

Results, postprocessed making use of dedicated macros, are reported in Figure 9b. As an example, contours of velocity, temperature and mixture fraction extracted on the meridional plane can be generated. The evolution of integral quantities such as temperature and emissions are extracted on the flame tube meanline, while PrF and PtF at the combustor exit are elaborated

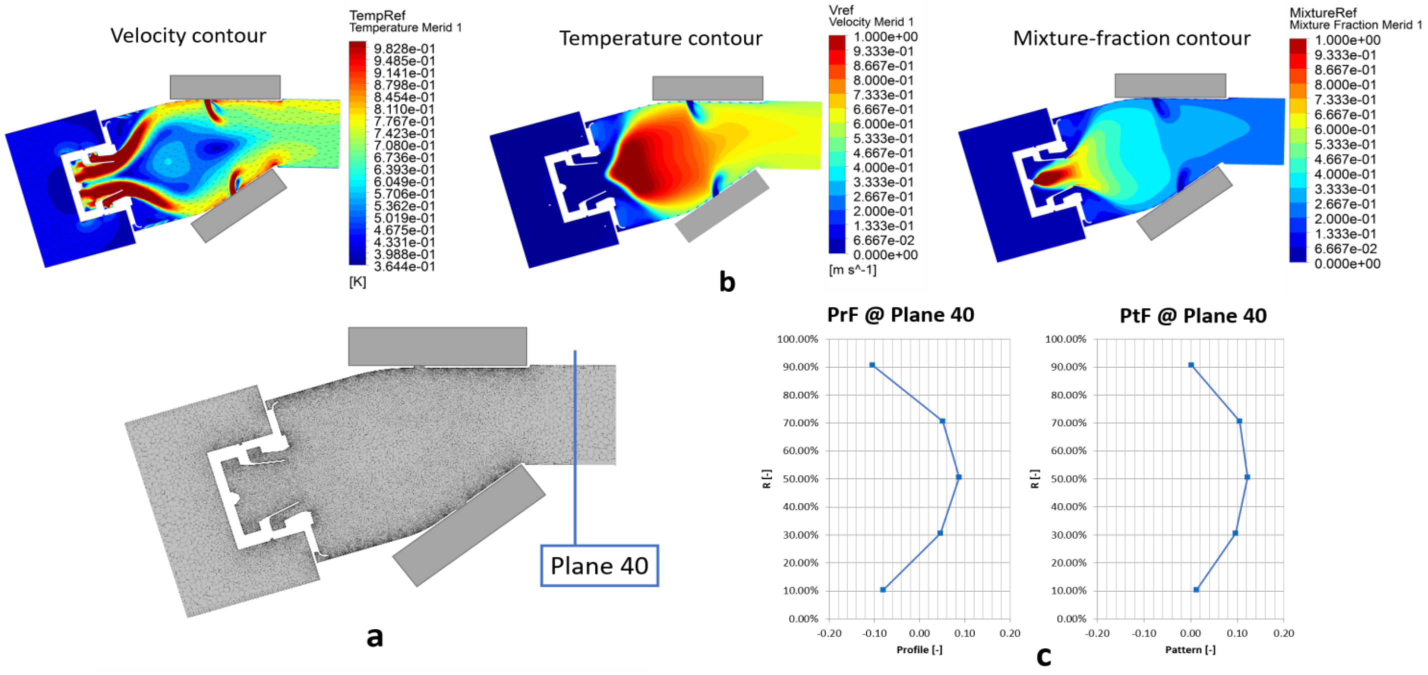


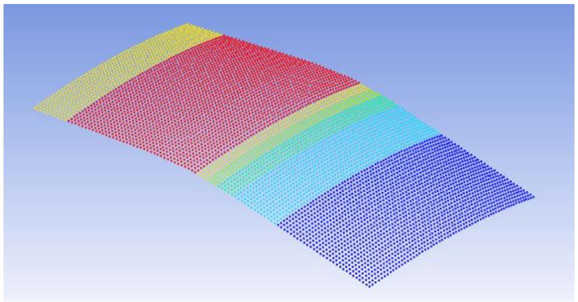
FIGURE 9: RESULTS OF MESHING IN THE PRE-PROCESSING STEP (a), MERIDIONAL CONTOUR-PLOTS (b) AND OUTLET TEMPERATURE PROFILES (c) IN THE POST-PROCESSING PHASE

and show in a 5-probe view (Figure 9c). Even if the validation phase is not completed, quantities of interest as well as parameters representative of combustor performance have been assessed. Considering CFD models developed in the scope of other projects [9, 12, 13, 14] and test data for the given test-point, good agreement has been found which provides confidence in the results presented in this work.

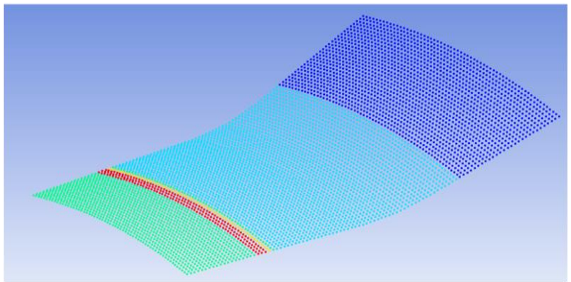
2.3 Thermal FE results

The maps shown in Figure 10 are obtained by a first update of the Therm-1D results with near-wall quantities (e.g. turbulence kinetic energy and local fluid velocity) exported from the previous CFD run.

For each liner, two representative 1D thermal network -one for centreline and one for mid-cup- have then been considered for generating a cloud of 1800 points: 60 axial-wise by 30 span-wise. The law to interpolate the linear distribution of thermal data was chosen to be the same for all the quantities of interest and in all liner locations. For the present case, a linear interpolation from ±10° to ±7.5° and polynomial from ±7.5° to 0° has been used. However, the procedure allows full flexibility in specifying different interpolation rules.



Outer Liner – Cold Side Convection



Inner Liner – Hot Side Convection

FIGURE 10: THERMAL BOUNDARIES FOR LINERS, READY TO BE MAPPED ON A 3D FEA MODEL.

CONCLUSIONS

In the present work, the Combustor DSI methodology has been described in the context of the aero-thermal preliminary design process adopted in *GE Avio*. The strategy exploited to enable the inter-tool and intra-tool automation is embedded in the tools developed for the procedure's functioning, which carry out:

- CAD generation
- CFD analysis
- Thermal FE evaluation

Developed in *ANSYS* environment, those wizards consist of ACT extensions that customize the product's functionalities through dedicated scripts.

To demonstrate its functioning, the NEWAC combustor concept has been processed within the DSI methodology. First, the test-case geometry was re-created, highlighting the steps in which this process unfolds. The outcome of this phase, a simplified fluid domain of the Combustor Discharge Nozzle (CDN), was automatically meshed. Then, a journal file with all the settings necessary for performing a CFD analysis (reacting RANS with FGM combustion modelling) was prepared and then executed. Results of the run were post-processed and relevant quantities and output reported: velocity, temperature and mixture contours, as well as emission distribution and PrF/PtF. A couple of exports for the thermal workstream were extracted. The first contains the wall conditions at the dome/injector location, which are directly mappable, while the second are the liner's Turbulence Kinetic Energy and Near-wall fluid velocity for updating the Therm-1D run. For the latter step, 3D thermal maps were so generated by means of the interpolation tool. Also, those maps were shown as scaled for different mission points, a functionality that enables the thermal transitory assessment.

To provide a measure of the time saved, an MBPM has been adopted. It can be said that by using the procedure, the resources can be reduced by 60% for a "new design" activity (intended for a design from scratch) and by up to 85% for a "re-design/re-run iteration" (respectively major or minor geometrical modifications to an already designed and/or different formulation of the boundary conditions problem). A further enhancement is given by rationalizing the information across the design process, an objective pursued via standardization of the files exchanged between the different tools. Such reduction, which means not only a shorter time frame for the design loop but also a more reliable process, results in more resources being available for the design space exploration. Indeed, the next steps are aimed at exploiting the aero-thermal tools in an optimization design methodology targeting aero parameters such as emission levels and PRF/PtF.

Following this demonstration phase, a validation of the NEWAC test case will be the subject of future papers and performed with test cell data and highly resolved computations.

ACKNOWLEDGEMENTS

The Combustor DSI is a research project funded by the Italian National Plan "Industria 4.0", aimed to digitize *GE Avio*'s combustor design processes. *ANSYS* and *DIEF* are partners in the development of this procedure.

A special mention goes to Pablo Aguado, Field Engineer at *ANSYS Spain*, Niccolò Casini and Riccardo Indelicato, Master Thesis Students at *University of Florence* and *Politecnico di Torino* respectively. My warmest gratitude goes to Antonio Peschiulli and Prof. Bruno Facchini for this exciting opportunity. This paper wouldn't be here without their precious contribution.

REFERENCES

[1] Dudebout, R., Reynolds, B., and Molla-Hosseini, K., 2008, "Integrated Process for CFD Modeling and Optimization of Gas Turbine Combustors," American Society of Mechanical Engineers Digital Collection, pp. 679–686.

[2] Gessel, M., Pfitzner, M., and Eggels, R., 2015, "Development of an Advanced Automated Knowledge Based Combustor Preliminary Design Process Suitable for Low NOx Combustion Systems Optimization," American Society of Mechanical Engineers Digital Collection.

[3] Pegemanyfar, N., Pfitzner, M., Eggels, R., von der Bank, R., and Zedda, M., 2008, "Development of an Automated Preliminary Combustion Chamber Design Tool," American Society of Mechanical Engineers Digital Collection, pp. 327–336.

[4] Pegemanyfar, N., Pfitzner, M., and Surace, M., 2009, "Automated CFD Analysis Within the Preliminary Combustor Design System PRECODES Utilizing Improved Cooling Models," American Society of Mechanical Engineers Digital Collection, pp. 289–300.

[5] Pegemanyfar, N., and Pfitzner, M., 2009, "State-of-the-Art Combustor Design Utilizing the Preliminary Combustor Design System PRECODES," American Society of Mechanical Engineers Digital Collection, pp. 465–475.

[6] Toal, D., Zhang, X., Bressloff, N., Keane, A., Witham, F., Gregory, J., Stow, S., Goddard, C., Zedda, M., and Rodgers, M., 2014, "Prometheus: A Geometry-Centric Optimization System for Combustor Design."

[7] Zhang, X., Toal, D. J. J., Keane, A. J., Witham, F., Gregory, J., Ravikanti, M., Aurifeille, E., Stow, S., Rogers, M., and Zedda, M., 2016, "Isothermal Combustor Prediffuser and Fuel Injector Feed Arm Design Optimization Using the Prometheus Design System," J. Eng. Gas Turbines Power, **138**(6).

[8] Rogero, J., and Rubini, P., 2003, "Optimisation of Combustor Wall Heat Transfer and Pollutant Emissions for Preliminary Design Using Evolutionary Techniques," Proc. Inst. Mech. Eng. Part J. Power Energy, **217**.

[9] Gordon, R., and Levy, Y., 2005, "Optimization of Wall Cooling in Gas Turbine Combustor Through Three-Dimensional Numerical Simulation," J. Eng. Gas Turbines Power, **127**(4), pp. 704–723.

[10] Angersbach, A., Bestle, D., and Eggels, R., 2013, "Automated Combustor Preliminary Design Using Tools of Different Fidelity," American Society of Mechanical Engineers Digital Collection.

[11] Ertas, A., 2018, *Transdisciplinary Engineering Design Process*, John Wiley & Sons.

[12] Achten, H., and Joosen, G., "The Digital Design Process," p. 6.

[13] Ertas, A., and Jones, J. C., 1996, *The Engineering Design Process*, Wiley.

[14] "Survey of Multi-Objective Optimization Methods for Engineering | SpringerLink" [Online]. Available: https://link.springer.com/article/10.1007/s00158-003-0368-6.

[15] "Assessment of a Set of Numerical Tools for the Design of Aero-Engines Combustors: Study of a Tubular Test Rig | Turbo Expo: Power for Land, Sea, and Air | ASME Digital Collection" [Online]. Available: https://asmedigitalcollection.asme.org/GT/proceedings-abstract/GT2009/48838/421/344775.

[16] Martin, K., and Osterling, M., 2012, *Metrics-Based Process Mapping: Identifying and Eliminating Waste in Office and Service Processes*, CRC Press.

[17] Locher, D. A., 2008, *Value Stream Mapping for Lean Development: A How-To Guide for Streamlining Time to Market*, CRC Press.

[18] "CFD Aerodynamic and Reactive Study of an Innovative Lean Combustion System in the Frame of the NEWAC Project | Turbo Expo: Power for Land, Sea, and Air | ASME Digital Collection" [Online]. Available: https://asmedigitalcollection.asme.org/GT/proceedings-abstract/GT2010/43970/377/339198.

[19] "3D Coupled Approach for the Thermal Design of Aero-Engine Combustor Liners | Turbo Expo: Power for Land, Sea, and Air | ASME Digital Collection" [Online]. Available: https://asmedigitalcollection.asme.org/GT/proceedings-abstract/GT2016/49798/V05BT17A009/239718.

[20] Colantuoni, S., Bake, S., and Badet, J. P., "Low Emissions Combustors Development for New Aero-Engines Core Applications," p. 6.