ON THE COMPLEXITY AND THE HANDLING OF MULTI-X SIMULATIONS IN THE SECONDARY AIR SYSTEM

D. Woelki*, K. Becker*, M. Schuff*, S. Tabassum*

* DLR, Institute of Test and Simulation for Gas Turbines, Virtual Engine and Numerical Methods, Forschungsallee 1, Augsburg, Germany

Abstract

The secondary air system (SAS) has paramount impact on the safe operation, remaining service life and overall efficiency of aero engines. Due to the manifold interactions with other engine components, there is demand for effective consideration of the SAS in various processes along the engine life cycle.

1D fluid network models are the state of the art for modeling large SAS domains. They depend on the availability and accuracy of characteristics of the fluid network's components and are considered as a low-fidelity tool when compared to the high-fidelity of 3D simulations, e. g. CFD or thermomechanical CSM. Since some SAS elements are very sensitive to changes in geometrical parameters or operation conditions, the widespread applicability of 1D models depends on an acceptable accuracy of the characteristics, which are build through experiments or high-fidelity simulations. For this reason, novel approaches are needed to combine 1D models with high-fidelity simulations in cost- and time-efficient ways.

This paper contributes to this demand by pointing out, which sensitive components are challenging in 1D fluid network modeling and should particularly considered in high-fidelity simulations. It is presented, how the 1D and 3D models complement each other and what it is needed to combine them in so-called Multi-X simulations (multi-fidelity, multi-discipline, multi-physics...). Furthermore, it is shown how Multi-X simulations can be managed in a consistent way, for which an ontology based data model named simulation topology is one key feature. This is complemented by a data management system which, among other things, ensures the efficient handling of large amount of data.

Keywords

secondary air system; multi-fidelity

NOMENCLATURE

Abbreviations

API	Application programming interface
CAD	Computer-aided design
CFD	Computational fluid dynamics
CSM	Computational structural mechanics
DMS	Data management system
FSI	Fluid-structure interaction
HPT	High-pressure turbine
LES	Large eddy simulation
Multi-X	Multi-fidelity, -discipline, -physics, -domain
PSN	Preswirl nozzle
RANS	Reynolds-averaged Navier-Stokes
REST	Representational state transfer
SAS	Secondary air system
TCP	Taylor-Couette Poiseuille
UUID	Universally Unique Identifier
WALE	Wall-adapting local eddy-viscosity

1. INTRODUCTION

The secondary air system (SAS) is indispensable for safe engine operation. Most of the secondary air is distributed to different stations in the turbines. There it is used for cooling of disks and blades and for sealing against hot gas ingestion from the primary gas path to inner cavities. The blade cooling design requires a specific combination of mass flow and temperature of the coolant. Sealing applications depend on local pressure and swirl. Along the paths of secondary air between its extraction points in the compressor (sources) and its return stations to the primary gas path (sinks), there is a significant change in pressure, temperature and swirl.

In the flow paths, the pressure ratio between their inlet and outlet stations is the main driver of secondary air mass flows. On the one hand, mass flows are mainly adjusted by so-called flow restrictors, which have a resistance to flow reflected in pressure drop. On the other hand, with the presence of swirl in the inner, rotating SAS, the static pressure is a function of the radius, depending on the characteristics between free and forced vortex. Moreover, fluid inside rotating components or close to disk walls are subject to windage, making the interaction between pressure and swirl distribution in the SAS complex. Due to windage, the fluid temperature also increases.

The SAS can be described as network of flow paths which branch or merge in cavities or junctions that are subsequently referred to as nodes. Every flow path is characterized by one or more elements, which affect one or more of the before mentioned parameters (pressure, temperature, swirl). Due to these changes of flow conditions,

doi: 10.25967/650079

adjacent flow paths interact with each other through the nodes.

1.1. Modeling the SAS as fluid network

The state of the art in modeling these networks of SAS flow paths are 1D models. This is primarily motivated by three facts: First, geometries of entire SAS domains are too large to handle them efficiently with high-fidelity methods like 3D CFD (computational fluid dynamics). Second, the flow characteristics of many elements in the SAS are relatively well known, which makes modeling with CFD dispensable. This particularly applies to flow restrictors like orifices, bores, or pipes, bends and junctions. This is completed with correlations or maps for widely-used SAS components like labyrinth seals. Therefore, 1D fluid network models are built from (usually) zero dimensional elements, which are defined by these characteristics. Some of the element types are geometrically parameterized. The individual elements are connected with each other in the flow paths and span, together with the nodes, the network. Given the thermodynamic boundary conditions in the sources and sinks of the SAS as well as rotational speeds, the simulation of a steady state operating point is performed with an iterative process with walltimes of typically few seconds on common desktop PCs - which is the third and final argument for these 1D models.
In this way, 1D models are mainly used for the computation of flow conditions in the boundaries of the SAS (sources and sinks) as well as in its inner nodes. The results, i. e. mass flows, temperatures and inner SAS pressures, are not solely required for the evaluation of the engine cycle (performance) and blade cooling. They can rather be used as boundary conditions for any simulation that depends on secondary air conditions, e. g. CFD in SAS components, thermal CSM (computational structure mechanics) and indirectly for life time estimation of blades and disks.
Shortcomings of 1D fluid network models are found in the limits of the characteristics, which represent the elements. These limits range from operation out of calibrated range to high sensitivity or complexity of the elements to the simple lack of any characteristics for some element types. Larger changes in the flow fields of certain types of cavities cannot be directly considered by the methodology of 1D models because the thermodynamic conditions in nodes and inlet/outlet planes of elements are abstracted to scalar quantities. Thus, new characteristics or correction data from experiment or high-fidelity simulation are needed.

1.2. The need for effective coupling of 1D and 3D

Due to their benefits, 1D SAS models can be used in many stages of the engine life cycle. Their application during detailed component design or in the scope of part concessions in the production process is already state of the art. Engine manufacturers' in-house 1D fluid network models are usually well calibrated and provide a relatively high level of detail, which is also accompanied with large numbers of nodes and elements. Even though not state of the art, these calibrated models could be important in the scope of engine health monitoring. In addition, the integration of 1D fluid networks to engine predesign is also not established. In contrast to later stages of the engine life cycle, predesign requires rather generic than detailed models. Nevertheless, the behavior of key components like the complex stator-rotor cavities of high-pressure turbines (HPT) must be relatively accurate in order to predict the shares and temperatures of secondary air in different sinks (blade cooling, sealing against hot gas ingestion) in critical operating points of a design flight mission. For this reason, knowledge gained from detailed component design will also improve the prediction accuracy of comparably lean 1D models in predesign. This, in turn, is not possible without high-fidelity simulations.
In summary, 1D fluid network models are the backbone of SAS simulation, but rely on appropriate input which is preferably generated through experimentally validated high-fidelity simulations. In return, the 1D models are suitable to provide appropriate boundary conditions for the high-fidelity simulations.
All of this leads to the demand of coupling 1D fluid network models with high-fidelity simulations of fluid and structures.

1.3. Multi-X simulations

For the further development of modern and future engine designs, it has for quite some time no longer been sufficient to consider components and/or disciplines independently in numerical terms. Some effects or problems only become obvious when they are considered in context, e. g., multiple components, several design levels, more than one domain, various disciplines, different aspects of a physical system, multiple time or distance scales and any combination of these. This paper provides a common approach to the formulation, execution and evaluation of multi-disciplinary, multi-fidelity, multi-physics, multi-scale or other complex simulations. For these different types of simulations the superordinate collective term Multi-X simulations is introduced.
This paper points out the need of Multi-X simulations in the SAS. Therefore, the focus here is on the treatment of CFD, CSM or FSI simulations in combination with 1D models. At first, examples for domains of the SAS are presented, that have particular impact to engine design and durability. It is also sketched how these domains are typically modeled in 1D fluid networks. Referring to one example of novel SAS concepts, namely a modulation of cooling air supply, the limits of 1D models and the demand for 3D simulations in consequence are pointed out in more detail. This is done by the discussion of a cavity in the SAS, which is sensitive to geometry changes in downstream or upstream flow paths. Together with two additional complex features that can be found in many SAS design, the high-fidelity models are presented, which will be applied in future Multi-X simulations. Finally, the challenges of these Multi-X simulations are discussed with emphasis put on several aspects of managing their high level of complexity in a still flexible manner.

2. 1D SAS MODELING

In this section, it is presented how the SAS is commonly split into different domains. These domains feature different types of elements, some of them characteristic for the respective domain. While the simulation of entire domains is commonly reserved to 1D fluid network models, the further breakdown to sub-domains leads over to those parts of the SAS where the application of high-fidelity models offers significant benefits.

2.1. Domains in the SAS

The entire SAS geometry spans over large areas of the aero engine. Secondary air is extracted at different interstages of the compressor at hub or tip radius. Most of the secondary air which is extracted at the hub to the inner SAS is used for

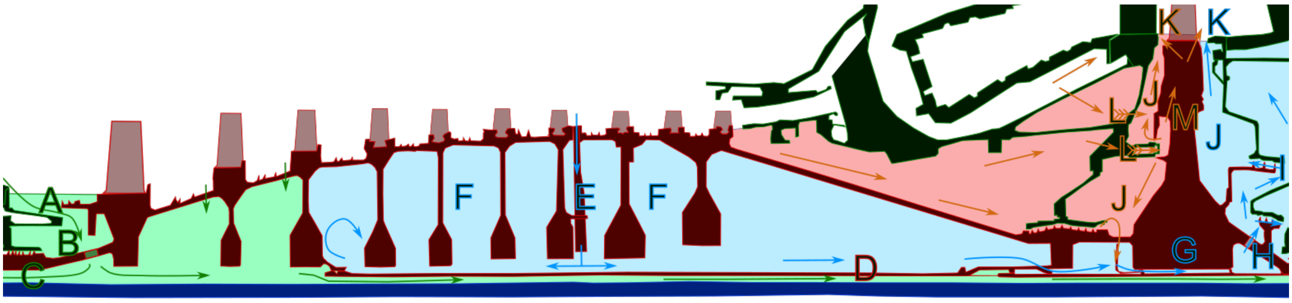


FIG 1. Domains in the inner, rotating SAS of the NASA EEE: high-pressure supply (red), mid-pressure supply (blue), low-pressure supply (green), based on [1]

cooling of rotating parts and sealing against hot gas ingestion while also contributing to other demands like balancing axial loads in the bearings. It is also used for sealing of the bearing chambers. All secondary air must be provided with sufficiently high pressure. Anyway, these pressure margins must be held low in order to keep the penalty on engine cycle level low. This is the primary reason for extracting secondary air from different compressor stages. The supply paths are separated from each other by walls. These zones of secondary air with different levels of source pressure may subsequently be named domains of the SAS.
Equally, the air which is extracted at casing side is defined by different pressure levels. In the outer SAS, the air is collected in stationary cavities and used e. g. for cooling and sealing of turbine casings or for use in external systems. It is furthermore common that secondary air at relative low pressure is transferred through pipes to the inner SAS where it is also used for sealing purposes at bearing chambers or in low-pressure turbines.
Figure 1 gives an example for different domains in the inner SAS of the NASA Energy Efficient Engine in the specification of Pratt & Whitney (NASA EEE) [1]. Since all secondary air is subject to pressure losses along its supply paths between sources and sinks, domains are strictly referred to their source pressure. Furthermore, it is not unusual that some domains are connected with each other as can be seen in the HPT disk bore region of NASA EEE where air from the inner high-pressure supply domain is merged into the inner mid-pressure supply domain.
There is no universal blueprint for the SAS or its domains. SAS design and so its geometry may significantly differ from engine family to engine family, and also within one engine family, there can be differences which make the individual designs largely incomparable.

2.2. Break down of domain features to element types

The SAS domains are usually modeled with each one independent fluid network. When domains are connected, the individual models must be matched in their interfaces, but initially, this takes complexity from the domain models while some of them can still contain more than a hundred elements.
Some of the typical elements are directly visible in Fig 1. In order to carve out the differences between generally well-understood and challenging types of elements, it is worth introducing the three illustrated domains. The modeling and characteristics of the more common elements are documented in the respective literature ([2], [3], [4], amongst others).

2.2.1. Low-pressure supply domain

The low-pressure supply domain is fed by air from the high-pressure compressor inlet. It passes gaps upstream and downstream of the inlet guide vane (A) and flows towards the low-pressure shaft. On its way to lower radii, it passes bores in the high-pressure rotor drum (B), which can be modeled as rotating orifices. Due to the change of radius and the rotor's influence, the pressure changes according to the known vortex laws of free, forced and intermediate vortices. One portion of this domain's inlet flow is used as sealing air in the front bearing chamber (left path, C) while the rest is transferred to the rear bearing chamber located under the low pressure turbine (not visible, right path). Most of the the latter flow path is characterized by a relative large annular gap that is located in-between the low-pressure shaft and a tube attached to the high-pressure rotor. This tube (D) separates the flow from the mid-pressure domain. This annular flow situation will be subsequently called flow in-between co-rotating shafts.
The fundamentals of flows through orifices are well understood, but the application of appropriate discharge coefficients is decisive. Although co-rotating shafts can be modeled as rotating ducts, they have been rarely investigated in the past [5]. In consequence, more accurate modeling of those co-rotating shaft flows in 1D fluid networks is desirable. This is not solely demanding enhanced computation of pressure losses (and hence mass flows), but also of the swirl component and heat transfer.

2.2.2. Mid-pressure supply domain

The mid-pressure supply domain is fed from a stage at higher pressure. In order to avoid the negative effect of a free vortex, tubed vortex reducers (E) are used for the transfer to the before-mentioned tube that separates the two domains (D). The air passes a number of rotating cavities of the high-pressure rotor drum (F) and flows around the cob of the HPT disk (G). It is purged through rotating bores to rotor-stator cavities in-between the HPT and the static structure mounting the rear bearing chamber (H). Further downstream, it passes labyrinth seals (I) and additional orifices and gaps. For labyrinth seals, characteristics are available from literature [6].
On the one hand, the challenges are once again found in accurate modeling of the swirl propagation in the rotor-stator cavities. On the other hand, this domain introduces two new complex types of SAS elements.
The first type refers to the outwards closed rotor cavities (F), each to be found between two compressor disks. Their impact on the provision of mass flow to the turbine can be considered as relatively low because the main flow is in the (near-tube) core region and only little dissipation is expected

when passing the disk bores of the last two compressor stages. Anyway, the near-wall fluid within these rotating cavities is pumped outwards and generates vortex structures in the cavity's core region. There, the fluid is transported back to the inner radius. Again, this results in heat transfer that has also impact on the thermal stabilization of the high-pressure system's structures.
The second type is the interface between the SAS domain and the primary gas path (K) downstream of the HPT rotor blade. This flow prevents ingestion of hot gas into the rotor-stator cavities (J). This is supported by sealing configurations subsequently referred to as rim seals. For 1D modeling, there are some low-fidelity approaches available, but accurate modeling of these important sinks of secondary air requires specific characteristics. This is due to the complex interactions between three major mechanisms which is described in [7].

2.2.3. High-pressure supply domain

The high-pressure supply domain features different flow elements and a relative large number of flow paths. Here, the flow restrictors are orifices and labyrinth seals, furthermore pre-swirl nozzles (L) and so-called disk receiver holes (M) [8], knife seals and different leakages in the region of the rotor blade's fir-tree. When geometry and characteristic quantities like inlet or outlet losses are known, these elements are basically straight-forward in modeling. Nevertheless, the accuracy of 1D fluid network models partly still relies on calibration data from experiment or high-fidelity simulation.
Since the cooling of the HPT demands for the highest share of secondary air, it is worth paying special attention to this domain. In particular, the HPT rotor cooling supply features another complex element, which again is a rotor-stator cavity, but combined with the impingement of secondary air through pre-swirl nozzles (PSN). These cavities are typically defined by each more than one inlet and outlet flow. The interaction of these flows can form complex flow structures. This is caused by the imposed mechanisms of pre-swirled air and windage including the heat transfer from disk walls to the cooling air. This finally results in fluid-structure interaction (FSI) that affects the seal clearances at inlet and outlet flows.

2.2.4. Splitting domains into sub-domains

Now giving the introduced term of the domain a clear meaning: Each domain shall be defined as discipline dependent control volume of a geometry to be modeled. In this way, the SAS is divided to fluid domains. Furthermore, a domain owns boundaries that are characterized as fluid ports (between two fluid domains) or surfaces (between one fluid and one structure domain).
In this way, the term domain is not reserved to complete SAS domains defined from sources to sinks (fluid ports). A domain can rather represent a single SAS element as well, or a combination of elements.
The formerly presented SAS domains are suitable to split the entire SAS in different fluid network models. It has also been outlined that some of the elements or even cavities, which can be defined as groups of elements, are worth for simulation with high-fidelity methods. In consequence, it is reasonable to now break the domains down into sub-domains which can be associated to certain high-fidelity models.
An example shall be given for the high-pressure supply domain, see Fig 2. In this figure, the cavities and elements

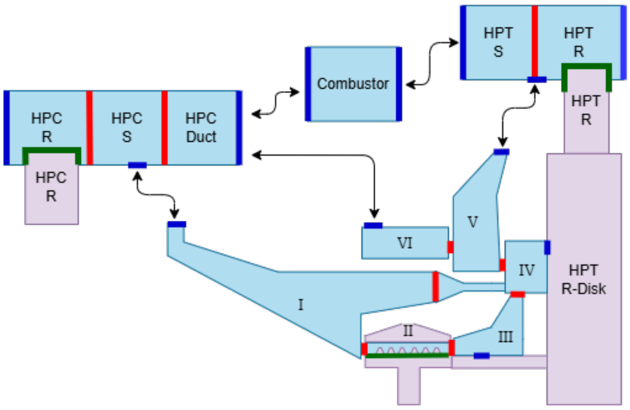


FIG 2. Inner high-pressure supply domain of the NASA EEE divided into sub-domains

from Fig 1 are now divided to sub-domains as following. Sub-domain I and VI are the two plena which contain the domain's sources of secondary air. The latter are the boundaries of the high-pressure rotor supply domain (blue color). The stepped labyrinth seal is set as sub-domain (II) which would allow the replacement by a CFD simulation or another kind of fluid simulation. Downstream, III represents the stator-rotor cavity which contains the air flow mixed to the intermediate-pressure rotor domain. Again, the fluid port to the inward domain is colored blue, while the fluid ports and interfaces between the sub-domains are marked red. Sub-domain IV contains both the main PSN and the short rotor cavity for blade coolant supply. For the sake of clarity, the outlet fluid port of IV to the disk receiver hole is also set as boundary of the entire domain. The stator-rotor cavity of sub-domain V is fed by leakage from IV and pre-swirled air from VI. It ends at the rim seal, which marks the boundary to the domain of the primary flow path in the HPT.

3. THE NEED OF MULTI-X SIMULATIONS

3.1. On the required level of detail

The state-of-the-art usage of 1D fluid network models is found where relatively much knowledge of the real SAS geometry is available, e. g. in detailed component design or attending the production process. Beyond, the models are basically also applicable for predesign [9] or concept studies for the integration of novel SAS features [10]. Furthermore it is worth to investigate whether detailed SAS models can be beneficially applied in the context of engine health monitoring, that is primarily based on engine performance without detailed analysis of interactions between the SAS and deteriorated turbo-components.
The challenges to integrate 1D SAS models to these applications are different:

- Conceptual engine design requires generic and lean models which can predict sound benchmarks of secondary air mass flows and temperatures. The models must consider the key elements of the SAS while e. g. detailed leakage flows are out of scope at this stage.
- Engine health monitoring is likely to require very detailed models like they are typically available after component design. This is because the progress of deterioration is slow, i. e. changes in SAS boundary conditions caused by the turbo-components or deterioration of SAS elements can usually only be reliably identified after multiple flight cycles. Thus, the fluid network models must be sensitive.

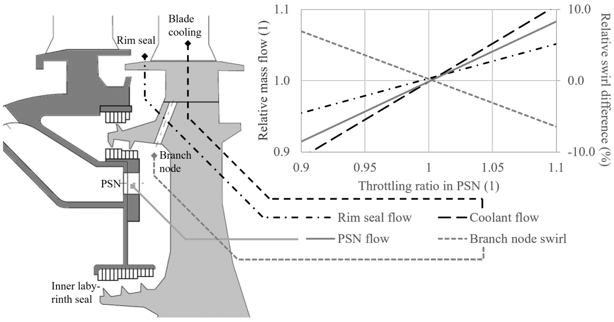


FIG 3. Relative secondary air mass flows at different throttle settings of pre-swirled supply flow and impact on swirl at cavity outlet, trend lines, based on [10] (CC BY 4.0)

- If concept studies on SAS features should provide quantitative trends, the use of detailed models is also indispensable because the integration of novel features is likely to cause a sensitive response of the SAS.

Both types of models - lean and detailed - can take essential benefit from high-fidelity simulations.

- In lean models, parameterized characteristics can be improved by or newly created from high-fidelity simulations.
- In detailed models, the used characteristics may get inaccurate when the operating conditions significantly differ from normal conditions for which the characteristics have been created. Also, the prediction gets inaccurate when the SAS geometry changes. The latter is always the case when new features are integrated and also when SAS elements deteriorate. This again requires correlations to correct the date characteristics.

Vice versa, the simulation of complex SAS elements requires accurate boundary conditions. These cannot always be scaled from engine performance data, and so 1D fluid network models are the best option to compute local boundary conditions for the high-fidelity simulations.

3.2. Example: Modulated cooling air

The limits of even detailed 1D fluid network models shall be explained with the help of one example. In [10], the potential of modulated cooling air in the supply of a HPT rotor has been investigated. This was done with a detailed 1D fluid network model of the respective SAS domain, calibrated to data from the reference engine. The major supply path of the high-pressure rotor domain, which is the pre-swirled flow injected to the stator-rotor cavity, was modulated by a hypothetical throttling of the PSN.

Apart from the impact of flow modulation on system level (i. e. engine cycle), the most fundamental effects regarding the flow in the cavity are summarized in Fig 3. There, the throttling ratio describes the flow modulation setting, i. e. the ratio between the pre-swirl nozzle's area at a certain throttling level referred to the nominal area (without flow modulation). On the one hand it can be seen that the relative mass flows at the rim seal and the rotor blade cooling are diverging at flow modulation. In other words, the distribution of secondary air within the domain changes. This is caused by a relative change of pressure in the branch node of the flow paths of cooling and rim seal air (upstream of disk receiver hole) [10]. This is likely to have fundamental impact on the flow characteristics of the outer part of the cavity: The 1D fluid network cannot consider the changes of the cavity's vortex system in the axial-radial plane. Since the second inlet flow of the cavity, located at the inner labyrinth seal, is also influenced by the altered flow through the PSN, it must be assumed that the flow topology of the cavity's inward region also changes.

Additionally, the near-disk swirl in the outer part of the cavity is also altered when the ratio between the inlet mass flows through PSN and inner labyrinth seal changes. Besides its potential impact on the three-dimensional flow field in the cavity, the change in swirl as well as the change of mass flow itself will also affect the heat transfer with the rotating disk. Thus, there is a demand for detailed, high-fidelity simulations beyond the fluid domain: It is rather required to investigate the interactions between fluid and structure.

- The change of disk cooling results in changed disk material temperatures, which has impact on disk lifetimes.
- Changes in the temperature of both disk and stator structures vary the thermally induced displacements, which is particularly important in instationary operating conditions. Since the flows through noncontact seals highly depend on gap width, there is mutual, bidirectional fluid-structure interaction.

For this reason, the full benefit and hence proof of concept for novel SAS features like modulated coolant supply still requires additional information from both CFD and thermal CSM models.

4. HIGH-FIDELITY SIMULATIONS ON SENSITIVE SUBDOMAINS

In order to enhance SAS simulation at the DLR Institute of Test and Simulation for Gasturbines, the following subdomains have been chosen for modeling with high-fidelity methods:

- HPT cavities with supply of pre-swirled rotor cooling air with two focuses: First, investigations on the sensitivities of the cavity's flow field, and second, fluid-structure interactions considering adjacent air seals.
- Co-rotating shafts
- Rotating compressor drum cavities

This section is dedicated to introduce how these subdomains are defined in the high-fidelity models, which specific type of simulation is to be performed and what boundary conditions are to be exchanged when coupled with a 1D model.

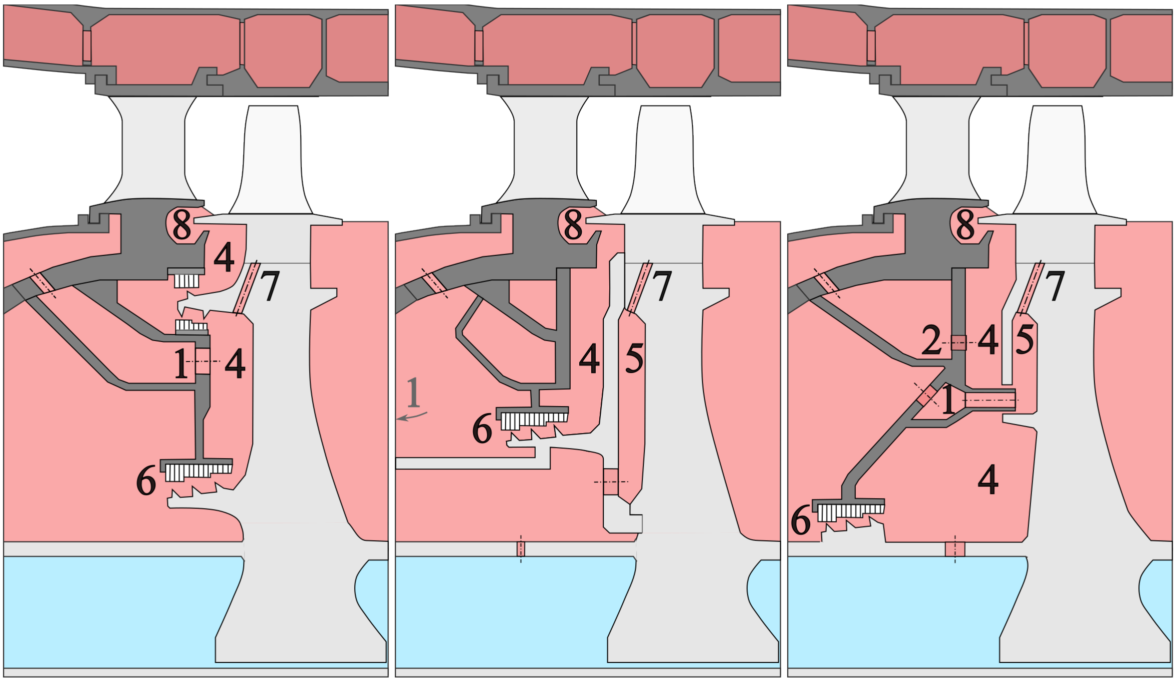
4.1. Rotor coolant supply with pre-swirled air

In preparation of the modeling of HPT cavities, Fig 4 provides insight to different designs for HPT rotor coolant supply. Although all three cases contain significant differences, most of the key features can be found in all of them. First of all, the highest share of cooling air is usually pre-swirled when being impinged from the stator to the rotor. This is done with pre-swirl nozzles in the stator structure (1), which is in the case of Fig 4b outside of the illustrated area.

Furthermore, there are always one or more inevitable flow paths between stator and rotor that contribute to the cooling of the disk at the one hand, but are also subject to windage on the other hand. These flows are mostly reduced by seals, e. g. labyrinth seals (6).

The actual, major difference between the presented base designs is how the coolant, which is provided to the disk receiver holes (7), is separated from the purge flow to the rim seal (8).

- In designs like illustrated in Fig 4a, there is a relatively large stator-rotor cavity (4). The coolant and rim seal flows branch at the cavity's outer radius. Due to the significant difference in wall velocities (stator vs. rotating disk) and especially due to the centrifugal effects of the near-disk



(a) No cover plate, adapted from [10] (b) With cover plate, adapted from [11] (c) Short rotating cavity, adapted from [1]

FIG 4. Different designs for provision of pre-swirled cooling air in HPT rotor

fluid, the flow topology is relatively complex and may lead to back flow to lower radii of the fluid in region of the stator wall [12].

- The second case (Fig 4b) separates the coolant and rim seal supply with an additional rotating structure, that builds a relatively long rotor cavity (5), while - in the very specific case - it is even an impeller. The rotating wall between (4) and (5) is sometimes referred to as cover plate. The swirl can be considered as fully forced vortex.
- In the third case (Fig 4c), the coolant is directly impinged by the pre-swirl nozzle (1) to a relative short rotating cavity (5). In opposite to the second design, there is leakage to the adjacent stator-rotor cavities. The stator-rotor cavity (4) of the rim seal feed flow holds similar features like the stator-rotor cavity in the first case, i. e. the leakage inlet flow (6) and a secondary PSN (2). The major difference is the absence of the feed to the receiver hole. Though not adapted in the very specific design of the second case, a secondary PSN could be another degree of freedom in SAS design.

In conclusion: Although SAS designs of the inner high-pressure supply domain may significantly differ, similarities can partly be found on sub-domain level. Even if the two subsequently introduced high-fidelity models for pre-swirled air supply start from only one specific design (Fig 4a), it is aimed to keep the modeling approach as general as possible in order to facilitate their application to other specific SAS geometries.

4.1.1. Investigation of flow topology with CFD

The focus of this high-fidelity model is put on the flow domain of stator-rotor cavities with pre-swirl nozzles only. It aims to investigate the sensitivities of the three-dimensional flow field to thermodynamic boundary conditions as well as geometry changes in the cavity. The promised outcome is threefold:

1) enhanced, general understanding on the effects of key parameters on the flow field
2) transfer of findings to 1D fluid network simulations where sensitivities are expectected to be decisive, e. g. studies on modulated SAS or studies on deteriorated SAS elements
3) initial characteristics of the cavity flow fields for lean 1D modeling in predesign

Thus, the case has been developed for pure 3D CFD RANS simulations in order to investigate the general changes in the cavity's flow field. For this purpose, a parameterized CAD model has been developed, on which CFD models can be easily set up for different geometries of the cavity. The detailed approach is presented in a dedicated paper, see [13]. However, the target is to use this model for the reproduction of characteristic parameters in the cavity's flow field in order to support 1D fluid network models.

Hence, putting the model to the context of Multi-X, the fluid network provides scalar input parameters at the boundaries of the CFD domain, e. g. pressures, temperatures and swirl information. The thermodynamic quantities required at CFD boundaries depend on the actual solver, but in general, as long as the physical state is well-defined, any conversion is possible. With the results from RANS CFD, the elements representing the sub-domain in the 1D fluid network can be updated or - in the long run - provided with complete characteristics. This transfer of boundary conditions and characteristics is generally applied in subsequent CFD cases as well (Sec. 4.2-4.3).

For scenarios in which the 1D model is basically provided with good characteristics and is meant to run in standalone mode, it might still be possible to keep the cavity's CFD model in backup. For example, if given operating conditions would result in extrapolation of available characteristics, the simulation process could switch to a Multi-X mode in order to resolve those uncovered conditions. In the aftermath, the characteristics in the 1D model could be again updated with the new, extended data.

4.1.2. Fluid-structure interaction with focus on labyrinth seal

As introduced before, any changed flow topology will also result in changed interaction between fluid and structures. For the investigated cavity, a high and bidirectional interaction (two-way coupling) is expected because the change in material temperatures due to changed heat transfer has influence on the seals of inlet flows, which are typically to be found at the lower radius of the cavity (see Fig 4, labyrinth seals (6)).
The current focus in this topic is the development of thermal CSM models of both the stator and rotor structures of a HPT rotor disk. In the context of Multi-X, it is planned to develop the interfaces between structure and fluid domains for both the high-fidelity model (3D CFD) and the low-fidelity model (1D fluid network).
In perspective, it is also planned to connect this case with the parametric model described above.

4.2. Co-rotating shafts

The flow between rotating shafts as mentioned in Sec. 2.2.1, is crucial to be analyzed with high-fidelity techniques as the heat transfer rates associated with the interaction of axial and rotational flows are complex and not sufficiently studied. The effects of heat transfer are particularly relevant in the low-pressure turbine, as changes in the temperature need to be estimated accurately. Although the interaction of co-rotating shafts in the concept of adaptive cooling air is negligible, it is important to understand the pressure losses involved to optimize the secondary air supply in the mid-domain. As demonstrated in Sec. 4.1, the SAS design differs between engine types. This is particularly applicable to the integration of inter-shaft flows, as they may operate at significantly different boundary conditions due to their associated sources and sinks. Examples can be found when comparing, e. g., [11] with [1], where the inter-shaft flows are part of different SAS domains. The presented baseline CFD models can be flexibly adapted to all of these SAS designs.
The simulations are performed as a crucial step towards the analysis of flow physics in a rotating annular gap with high swirl parameters and thermal gradients at engine-representative conditions. The configuration represents a Taylor-Couette Poiseuille (TCP) flow that has complexities associated with lateral curvature effects, anisotropic turbulence modification, and three-dimensional turbulent boundary layers. TCP configurations are generally studied for fully developed flows that are simulated through axial periodicity [14,15]. This neglects the entrance effects where the thermal and hydrodynamic boundary layers are thin, thereby making the simulations independent of inlet swirl data and the length of the shafts.
The effects of rotation on the flow laminarization, the resulting heat transfer rates on the shafts, and the pressure losses are of interest. It is important to study how the flow field changes for the combinations of rotation rates within the operating profiles of shafts. The flow in the gap of the annulus is dependent on the ratio of the rotation rates of the shafts and the amount of throughflow of secondary air. Within the known literature, such configurations have rarely been investigated for the desired flow conditions. This makes it important to consider high-fidelity unsteady simulations like Large Eddy Simulation (LES) to understand the flow physics and to quantify the deviations and scrutinize the behavior of steady Reynolds-Averaged Navier-Stokes (RANS) simulations. Fig 5 shows the flow fields obtained through RANS

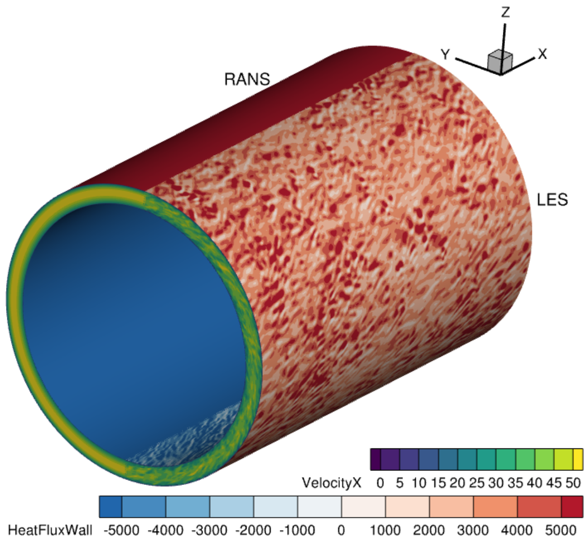


FIG 5. Heat flux on the rotating shafts and axial velocity distribution obtained for developed flow in annular gap through RANS and LES

and LES. While showing the discrepancies in the absolute heat flux obtained on the walls, the two modeling techniques complement each other, showing a qualitative agreement. The LES results show the near-wall vortical structures that modify the turbulence levels in the gap, thereby affecting the heat transfer rates.
The simulations may be extended to more realistic scenarios like shafts with finite lengths and specified inlet swirl. Additionally, it would be helpful to derive correlations to facilitate the estimation of pressure losses and heat transfer rates. These correlations obtained from simulations and calibrated with the experimental data are required as input to the 1D models of SAS that are state-of-the-art for different phases of gas turbine development.

4.3. Compressor drum cavities

Cavities formed by co-rotating disks and the outer cylindrical surface formed by the drum, called the shroud, are present in the compressor and the HPT. The transported secondary air at the low-pressure shaft creates an axial throughflow between the disks. The flow structures significantly influence the heat transfer rates within the disks and thereby influence the disk deformations. The disk thermal response with high temperature gradients has a direct effect on the blade tip clearances, which would then affect the engine performance [16]. Therefore, the behavior of the disk deformations and the component performance must be considered in the detailed engine design.
The disks and the shroud are generally heated in the steady engine cruise conditions, whereas the secondary air near the inner shaft is at relatively low temperatures. This temperature difference leads to instabilities in the flow, causing unsteady flow effects arising from buoyancy in the presence of centrifugal forces. The centrifugally-driven buoyancy convection is a heat transfer phenomenon that causes large-scale inertial structures that rotate within the cavity [17]. These large-scale structures, along with the interaction of disk Ekman layers and the axial throughflow, result in highly unsteady heat transfer rates in the disks and the shroud. Accordingly, unsteady simulations such as LES are generally considered appropriate to model these flows. Fig 6 shows the temperature distribution in the

mid-cavity plane obtained through LES-WALE simulation. The disks and the shroud are heated whereas the inlet temperature of axial throughflow is 298 K. Significantly localized radial inflows and outflows due to buoyancy result in non-axisymmetric temperature distribution in the cavity. This reflects in high-temperature gradients and non-uniform heat transfer rate distribution in the disks and the shroud. The interaction of inertial forces with buoyancy and rotating forces results in complex, unsteady, and three-dimensional flow structures. The buoyancy is, in turn, affected by the disk temperature distribution, which makes this a highly conjugate heat transfer problem. This also leads to significant inter-cavity interaction, where the flow structures are changed due to the conduction in the disks [18]. This interaction of conduction in the disks with the varying flow structures is not fully understood. Such configurations demand high-fidelity techniques where the flow physics and the resulting heat transfer are highly dependent on buoyancy effects, rotation rate, and throughflow mass flow rates resulting from the operating conditions.

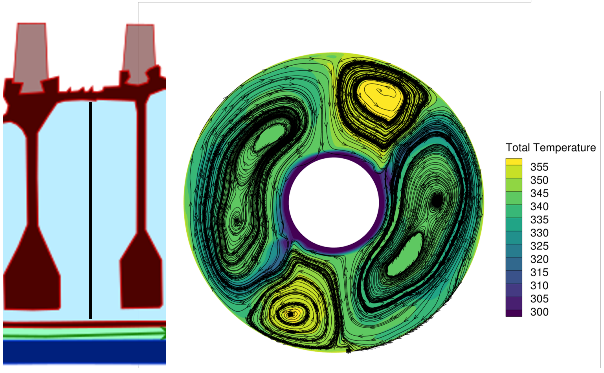


FIG 6. Temperature distribution in the cavity mid plane with the location indicated by line

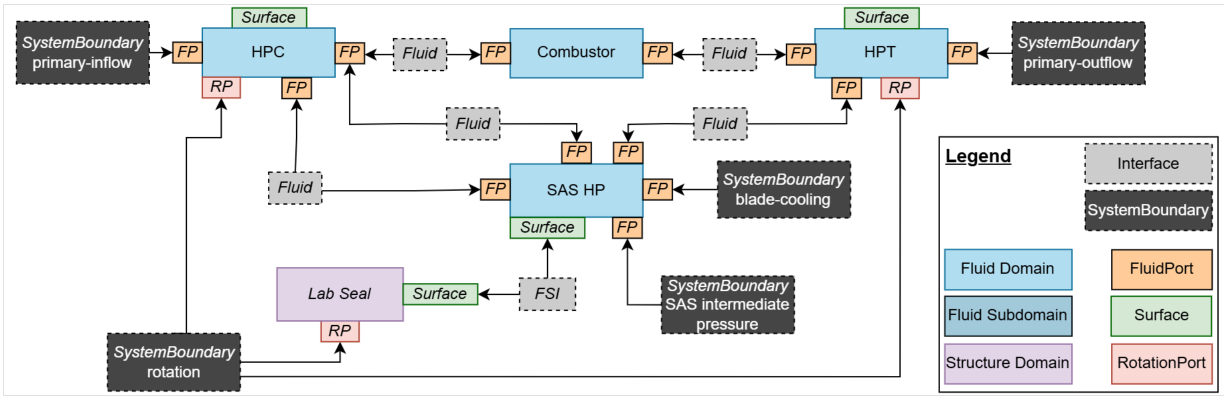
Modeling of these cavities is generally done with unsteady CFD simulations and experiments. Several correlations are obtained to predict the heat transfer rates on the shrouds, the disks, and the bore regions. However, due to the flow complexity and dependence of the flow structures on the operating conditions, these correlations should be continuously enhanced to match the data from high-fidelity simulations and measurements from the experiments. These methods have been used to investigate the flow in rotating cavities and have made notable progress. Essential here is to know the swirl of the fluid, the temperature distribution in the disks, and the temperature difference between the walls and the fluid. Accurate modeling of the flow at engine-representative conditions requires estimation of appropriate mass flow rate and rotation rate, preferably derived from 1D models. Enhanced correlations obtained from experiments and unsteady simulations can be used to feed the characteristics that represent the cavity domains within the 1D model. Besides modeling the heat transfer rates on the walls of the cavity, these correlations would also be useful to estimate the temperature changes in the throughflow across the cavity.

5. MANAGING MULTI-X SIMULATIONS

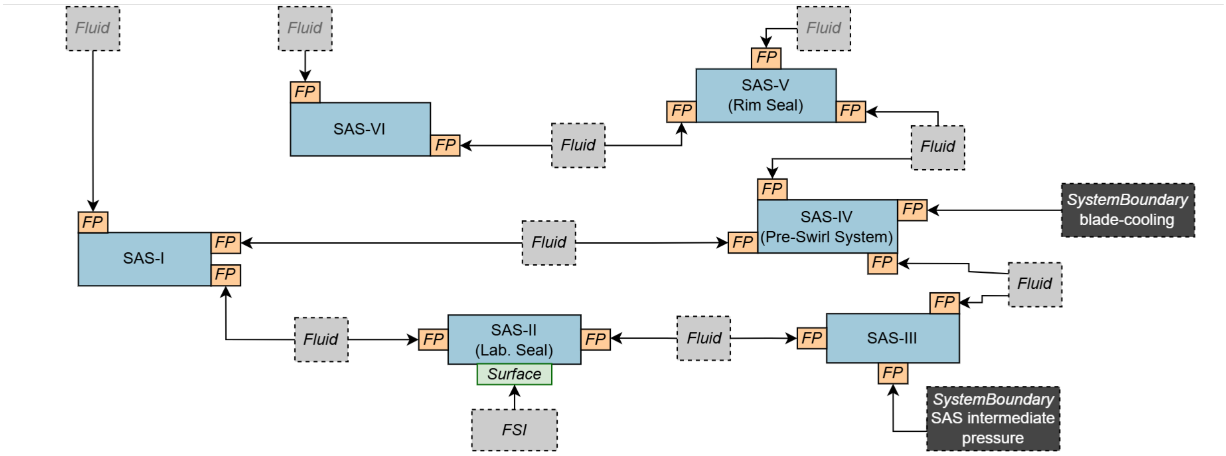
The broad use of simulations today, ranging from simple, short simulations to very large, complex simulations, demands intensive attention to their management. This not only includes FAIR (findable, accessible, interoperable, reusable) data management but also requires a deeper insight into the descriptions of the individual simulations and their processing. For a data-driven approach, a data model for the here described simulation area is needed.
As the focus shifts towards a collaborative approach to engineering work, the necessity for a detailed description of such simulations becomes ever more urgent. This will increase the already large amount of data even further, that is distributed across different systems and the various process steps in such a design approach. This data consists of different data sizes and results from the many different involved disciplines. A consistent data formulation is necessary across all phases, from initial design to detailed, high-fidelity component simulations.
Within DLR, the software GTlab (Gas Turbine Laboratory, [19]) is used as the platform for the so-called Virtual Engine [20]. The core of this software is open source and provides a common central data model. New features can be added through plugin libraries, which extend the data model and add specific workflows.
GTlab aims to include all above described phases from collaborative design into a framework, providing cross-disciplinary capabilities, as well as workflow orchestration [21, 22]. The intended use is a simulation- and data-based description of gas turbines, test rigs or parts of it over the whole product life cycle and in interaction with the respective environment.

5.1. Formal description of simulations

An ontology based data model named simulation topology was introduced by Schuff et al. in [23]. It is categorized into four subjects: domains and topology, boundaries and interfaces, boundary conditions, as well as results. A topology usually consists of more than one domain. In Fig 2, the topology comprises four fluid domains, named HPC, combustor, HPT and SAS, as well as two structural domains, a HPC rotor disk and a HPT rotor disk. The blue lines stand for interfaces between fluid domains whereas the green lines describe the interfaces between fluid and structural domains. The red lines mark the interfaces in a domain and separate its subdomains which has been already discussed in Sec. 2.2.4.
Each domain corresponds to a single-discipline simulation confined by its boundaries. The boundaries between adjacent domains are connected by interfaces. An excerpt of this topology is shown in Fig 7, focusing on the interaction between fluid and structure for the inner high-pressure supply domain of an engine. The (sub-)domains of the fluid and structure are drawn in blue and purple, respectively. Fluid ports, drawn as *FP* in orange, define permeable boundaries whereas surfaces describe impermeable boundaries, illustrated in green here. Rotation ports, drawn as *RP* in red, differ as they only apply a certain rotational speed onto the domain. The boundaries are connected by inferfaces (*Fluid* and *FSI*) drawn in grey. For the different combinations specific interfaces are required. The interfaces connecting two fluid ports follow the gas path and couple two fluid domains, as can be seen in Fig 1. The interfaces connecting two surfaces couple a fluid domain with the adjacent structural domains, thus representing a fluid structure interaction in this context.
Ontologies - and graph databases derived from them - typically encode explicitly named relationships between two entities, commonly represented as triples (subject-predicate-object). In the example presented here, those named relationships are left unlabeled as the intention of each ar-



(a) Component level view of the four domains HPC, combustor, HPT, and SAS HP



(b) Subdomains of the SAS HP

FIG 7. Formal description of the simulation topology of the inner high-pressure supply domain of the NASA EEE

row is considered as clear ("interface connects boundary"). Boundaries of each (sub-)domain are bound to this domain in a parent-child relationship, and therefore directly attached in the graph.

The primary gas path is divided into the domains HPC, combustor and HPT. The secondary gas path is completely represented through the inner high-pressure supply domain (SAS HP). The fluid part of the SAS splits into various subdomains as already described above. Note that this version is for the sake of simplicity in context of the paper and for demonstration only. A more realistic simulation topology of an aero engine's secondary air system will include several dozen to hundreds of individual elements in the secondary gas path. Not all of them might be confined to individual subdomains on the same level, as the simulation topology in general allows clusters or nested subdomains of subdomains.

In this example, the subdomains confine five separate regions in the SAS which may be represented through surrogate models, characteristics, or even CFD. The fluid interfaces to the primary gas path are grayed out. The treatment of these interfaces will be discussed below and depends on the individual simulation workflow and goal. The FSI at the labyrinth seal is the only connection to a structural domain. Realistically, for each subdomain the relevant modeling of an FSI (deformation, gap closing/opening, and/or heat transfer) must be taken into account. The same goes for the rotation ports of some of the subdomains. Technically, moving walls have to be addressed in the simulation of each subdomain.

For the sake of simplicity, rotation ports in the SAS fluid subdomains are omitted in this view.

The here presented description and its underlying data model are largely independent of the fidelity level and the used solver applied for the individual domains. The data model offers an abstract formulation to describe multi-domain computational problems and is the base to store detailed information of such simulations in a knowledge graph. The underlying ontology is derived from the W3C Prov standards [24] and the common aircraft engine model used by GTlab enabling to track information throughout the various fidelity levels, to monitor feedback between low- and high-fidelity analysis and to integrate this model in GTlab.

5.2. Performing multi-fidelity simulations

The integration of different solvers' implementations is separated from the formulation requiring, for example, specific interfaces, such as CFD-CFD (e. g. mixing plane or zonal), or 1D-CFD (e. g. interpolation/integration between 0D and 1D/2D values). Of course, the actual implementation is not trivial and largely depends on more factors, especially when interpolating schemes are to be utilized. On a coarse level, two solving strategies can be distinguished as laid out in [25]. The main distinction is whether solvers are carried out in sequential order or in parallel. The strategies put different focus on the aspects of how implementation and integration into the Virtual Engine platform is done.

5.2.1. Sequential solving strategies

The general approach of a sequential order is applicable in many engineering tasks. Each solver is carried out after the other, often by different disciplines or engineering teams. An interchangeability and standardized exchange of data at the interfaces is therefore paramount to avoid conversion or transfer errors by manual tasks.
In such cases, the interfaces to other (sub)domains act through result tables or files. In this way they behave like boundaries of the currently investigated (sub)domain, which have constant values prescribed from previous processes.
A prime example for this approach is the component zooming: high-fidelity simulations for a subdomain are performed and the results are used to build or correct characteristics and surrogate models. This approach allows for a fast analysis in cases where the behavior is investigated on a system level (e. g. performance analysis, changes due to operating condition), or the focus is on other parts of the system (e. g. re-design in a different region).
To give an example: The pre-swirl system, as described in 4.1, shall be characterized by CFD simulations. Inflow and outflow boundary conditions of the CFD solver are prescribed as appropriate, e. g. total condition at inflow, static pressure at outlet. The actual values stem from previous run of other tools like the 1D network solver.

5.2.2. Parallel solving strategies

A parallel approach might be utilized when direct or strong coupling effects of the different domains are of interest, e. g. coupling the performance synthesis of the primary gas path with the actual flow through the secondary air system, and thus getting a better prediction for the thermodynamic quantities of blade cooling supply. On the other hand, if the CFD simulations need accurate boundary conditions (e. g. strong coupling effects between mass flow and small changes in the fluid domain), a network model might be run in parallel and an exchange after several CFD solver iterations can be triggered, updating the CFD boundaries.
In such cases, all the involved domain solvers run in parallel and an exchange of the opposing boundaries is performed after several iterations. Depending on the connected interfaces, convergence and interpolation techniques can be chosen from classical numerical methods. When the amount of exchanged data, either because large field vectors are transferred, or because the exchange is repeated frequently, an in-memory exchange without file I/O might become beneficial.

5.3. Data management system

Executing such simulations requires a data management system (DMS) that supports efficient data handling, robust integrity guarantees and automated processing. Multi-X simulations generate files ranging from small size, e. g. for 1D studies, to extremely large size from high-fidelity simulations. The DMS must manage these files efficiently across transfer, storage and life cycle operations. It has to ensure the complete provenance of the simulations conducted so that individual process steps are reusable at any time and their data sets are fully searchable. Furthermore, the integrity for both individual files and compound data sets must be ensured. To minimize the transfer of large data volumes, the DMS should be able to perform simple computations in situ on data stored within the system. The system is used in complex, automated process chains and workflows as part

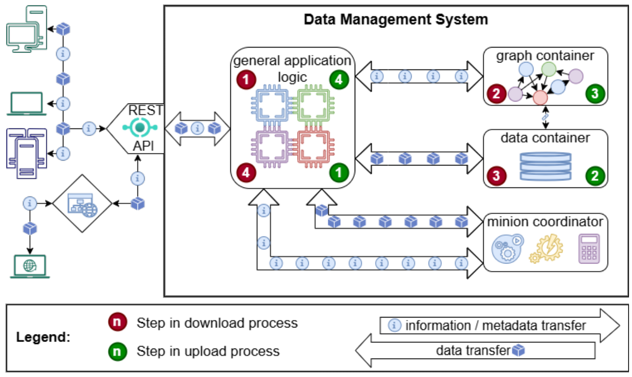


FIG 8. Concept of the central data management system

of a simulation framework like GTlab. Therefore, it must provide interfaces for seamless integration.
These requirements lead to a central DMS approach published in Haug et al. [26] following a single source of truth concept as shown in Fig 8. The main features of the central DMS are the graph database storing all metadata and provenance information as core of the application, the file management comprising different database storage systems (e. g. Shepard), an interface and main application logic exposed via a REST API (representational state transfer application programming interface) as well as an on-demand data processing to allow partial access to large files and to perform smaller evaluation tasks. Comparable to standard version control systems, this design is aimed to guarantee a comprehensive traceability of all data changes throughout the product life cycle.
The data is stored as a knowledge graph in the graph database using an ontology-based structure. An overview of this system is given in [26]. In contrast to a purely hierarchical structure, this approach can be viewed as a network with relationships between entities in any direction, being suitable for the complex simulations discussed. For each node in the graph database important properties have to be set being able to find the node and to identify its related element: the name, an Universally Unique Identifier (UUID), the type of the element (e. g. file) and a key-value list with optional further properties, depending on its type, (e. g. the path of a file in a file storage system).
In Fig 8 the process steps of two standard transactions are indicated by the colored circles with their corresponding number to demonstrate the usage of the DMS to illustrate the process flows. The red path shows the data and information flow to download a file from the DMS. Provided that the UUID of the file node is known, a request is sent to the DMS (①) that is forwarded to the application logic layer in the system. The application logic then queries the graph database to obtain the corresponding node and thus the file details (②). The following step is to download the file from the data storage (③). If the query includes further data processing tasks, these are now executed by a task manager (minion coordinator). Finally, the file and all other data generated by these tasks are made available to the user via the REST API (④).
The upload transaction follows the reversed order represented by the green path. First, a file is uploaded to the DMS via the REST API. The general application logic generates a UUID for this file (①). If data processing tasks are part of the query they are now executed. Any result is stored in the data storage along with the original file(②). In the next step (③), the information for the graph nodes is collected and forwarded to the graph database to be

embedded in the knowledge graph. In the final step, the UUID of the file is sent back to the user via the REST API to report successful completion (④).

6. CONCLUSION AND OUTLOOK

In summary, the engine simulation processes are converging on integrated, end-to-end workflows that are largely independent of the life cycle phase. From concept design through detailed development to fleet monitoring, there is a growing need for rapid, but accurate analysis with improved resolution at the subcomponent level. In many tasks, the SAS has only been considered on a coarse level in the past. Bringing both together - highly integrated simulations and the consideration of the SAS - it is obvious that SAS simulation requires both accurate and efficient approaches.

As presented, the toolbox of SAS simulation contains the fast 1D models for large domains and also requires high-fidelity simulations for complex elements or sub-domains. Both fidelity types are relevant and must complete each other. This can only be efficiently done by combining them in Multi-X simulations.

Nevertheless, it must be underlined that Multi-X simulations are not to be reduced to the simple coupling of two simulations. Even if this paper focused on fluid domains, i. e. 1D network models and 3D CFD, and pointed to the demand of FSI in SAS simulation, such coupling of two solvers is only a subset of Multi-X. While many principles of solver coupling is based on two domain coupling schemes, Multi-X extends these approaches to the coupling of many solvers, either in parallel execution or as sequential tools. The simulation topology provides a generic description of the interconnections of all participating domains.

Performing such simulations leads to large amount of data generated that has to be managed. The simulation topology presented provides the basis for making such simulations findable, accessible, interoperable and reusable. For this, an advanced data management system, such as the one presented here, has to be used.

As next steps, it is planned to demonstrate the here presented aspects of Multi-X to some of the presented SAS sub-domains, each involving high-fidelity and 1D fluid networks. For all cases, efficient strategies should be suggested for future adaption to industry. Furthermore, the DLR Institute of Test and Simulation for Gas Turbines aims to increase its capabilities in SAS simulation by also exploring additional complex sub-domains. One example is the simulation of rim seals as one of the critical type of sinks in turbines.

Contact address:

dominik.woelki@dlr.de

References

[1] Pratt and Whitney Aircraft. Energy efficient engine. Volume 1: Component development and integration program. NASA Contractor Report CR-173084, 1981.

[2] I. E. Idel'chik. *Handbook of hydraulic resistance. Coefficients of Local Resistance and of Friction.* The U.S. Atomic Energy Commission, 1960.

[3] D. S. Miller. *Internal Flow Systems*. BHR Group Limited, The Fluid Engineering Centre, Cranfield, Bedfordshire, UK, 1978. ISBN: 0-947711-77-5.

[4] K. J. Kutz and T. M. Speer. Simulation of the secondary air system of aero engines. *Journal of Turbomachinery*, 116(2), April 1994. DOI: 10.1115/1.2928365.

[5] F. Fuchs and L. Cordes. Secondary air system: A current review and future investigations. In *Proceedings of ASME Turbo Expo 2022*, 2022. DOI: 10.10115/GT2022-81262.

[6] H. Zimmermann and K. H. Wolff. Air system correlations: Part 1 - labyrinth seals. In *Proceedings of ASME Turbo Expo 1998*, 1998. DOI: 10.1115/98-GT-206.

[7] F. Gao, J. W. Chew, and O. Marxen. Inertial waves in turbine rim seal flows. *Phys. Rev. Fluids*, 5(024802), February 2020. DOI: 10.1103/PhysRevFluids.5.024802.

[8] M. Dittmann, T. Geis, V. Schramm, S. Kim, and S. Wittig. Discharge coefficients of a pre-swirl system in secondary air systems. In *Proceedings of ASME Turbo Expo 2001*, 2001. DOI: 10.1115/2001-GT-0122.

[9] D. Y. Kulkarni and L. di Mare. Virtual gas turbines part II: An automated whole-engine secondary air system model generation. In *Proceedings of ASME Turbo Expo 2021*, 2021. DOI: 10.10115/GT2021-59720.

[10] V. D. Woelki. *Modulated Secondary Air Systems for Enhanced Off-Design Operation of Stationary Gas Turbines and Aero Engines*. PhD thesis, Technical University of Berlin, Berlin, Germany, September 2022. DOI: 10.14279/depositonce-16063.

[11] E. M Stearns. Energy efficient engine core design and performance report. NASA Contractor Report CR-168069, General Electric Co., 1982.

[12] H. Karabay, J.-X. Chen, R. Pilbrow, M. Wilson, and J. M. Owen. Flow in a "cover-plate" preswirl rotor–stator system. *Journal of Turbomachinery*, 121(1), January 1999. DOI: 10.1115/1.2841225.

[13] L. Gante. Computational fluid dynamics simulation for a parametric preswirl stator-rotor system (submitted). In *Deutscher Luft- und Raumfahrtkongress 2025*, 2025.

[14] S. Poncet, S. Viazzo, and R. Oguic. Large eddy simulations of taylor-couette-poiseuille flows in a narrow-gap system. *Physics of Fluids*, 26:1070–6631, October 2014. DOI: 10.1063/1.4899196.

[15] M. Hadžiabdić, K. Hanjalic, and R. Mullyadzhanov. Les of turbulent flow in a concentric annulus with rotating outer wall. *International Journal of Heat and Fluid Flow*, 43:74–84, June 2013. DOI: 10.1016/j.ijheatfluidflow.2013.05.008.

[16] D. E. Bohn. How far have we been? - summary of investigations on rotating cavity at idg, rwth aachen university. Volume 3:489–497, December 2009. DOI: 10.1007/s11708-009-0040-y.

[17] D. E. Bohn, G. N. Deutsch, B. Simon, and C. Burkhardt. Flow visualisation in a rotating cavity with axial throughflow. Volume 3: Heat Transfer; Electric Power; Industrial and Cogeneration:V003T01A084, May 2000. DOI: 10.1115/2000-GT-0280.

[18] Z. Sun, F. Gao, J. W. Chew, and D. Amirante. Large eddy simulation investigation of low rossby number buoyant flow in rotating cavities. *Journal of Engineering for Gas Turbines and Power*, 144(12), October 2022. DOI: 10.1115/1.4055686.

[19] S. Reitenbach, J. Schmeink, M. Bröcker, M. Nöthen, and M. Siggel. GTlab, Aug. 2024. DOI: 10.5281/zenodo.13450248.

[20] DLR. Compute before flight. *DLRmagazine*, 170:10–13, May 2022. ISSN: 2190-0108. https://www.dlr.de/en/media/publications/magazines/2022_dlrmagazine-170-compute-before-flight.

[21] S. Reitenbach, M. Vieweg, R. Becker, C. Hollmann, F. Wolters, J. Schmeink, T. Otten, and M. Siggel. Collaborative aircraft engine preliminary design using a virtual engine platform, part A: Architecture and methodology. In *AIAA Scitech 2020 Forum*, 2020. DOI: 10.2514/6.2020-0867.

[22] S. Reitenbach, C. Hollmann, J. Schmeink, M. Vieweg, T. Otten, J. Häßy, and M. Siggel. Parametric datamodel for collaborative preliminary aircraft engine design. In *AIAA Scitech 2021 Forum*, 2021. DOI: 10.2514/6.2021-1419.

[23] M. Schuff, J. P. Haug, K. Becker, M. Siggel, J. Schmeink, and S. Reitenbach. Data management in a collaborative design architecture, part C: Simulation topology. In *AIAA Scitech 2025*, 2025. DOI: 10.2514/6.2025-1372.

[24] K. Belhajjame, R. B'Far, J. Cheney, S. Coppens, S. Cresswell, Y. Gil, P. Groth, G. Klyne, T. Lebo, J. P. McCusker, S. Miles, J. D. Myers, S. Sahoo, and C. Tilmes. PROV-DM: The PROV Data Model. Technical report, April 2013.

[25] M. Schuff. Towards multi-fidelity simulations in the context of the virtual engine. In *Deutscher Luft- und Raumfahrtkongress 2023*, 2023. DOI: 10.25967/610297.

[26] J. P. Haug, K. Becker, M. Schuff, M. Siggel, and S. Reitenbach. Data management in a collaborative design architecture, part A: The basic concept of the underlying data management system. In *AIAA Scitech 2025*, 2025. DOI: 10.2514/6.2025-1370.