Proceedings of ASME Turbo Expo 2014: Turbine Technical Conference and Exposition
GT2014
June 16-20, 2014, Düsseldorf, Germany

GT2014-25886

PROMETHEUS: A GEOMETRY-CENTRIC OPTIMIZATION SYSTEM FOR COMBUSTOR DESIGN

Xu Zhang[*], **David J.J. Toal**
Neil W. Bressloff, Andy J. Keane
University of Southampton
Southampton, UK, SO17 1BJ

Frederic Witham
Jonathan Gregory
Rolls-Royce plc
Bristol, UK, BS34 7QE

Simon Stow, Christopher Goddard
Marco Zedda, Mark Rogers
Rolls-Royce plc
Derby, UK, DE24 8BJ

ABSTRACT

The following paper presents an overview of the Prometheus design system and its applications to gas turbine combustor design. Unlike a traditional "optimizer-centric" method, Prometheus aims to reduce both the level of workflow complexity and rework by taking a more "geometry-centric" approach to design optimization by shifting the control of script generation away from the optimization program to the computer aided design (CAD) package. Prometheus therefore enables significant geometry changes to be automatically reflected in all subsequent scripts necessary for the analysis of a combustor. Prometheus' current capabilities include automatic fluid volume generation and aero-thermal and thermo-acoustic network generation as well as automatic mesh and computational fluid dynamics (CFD) script generation.

1 INTRODUCTION

Gas turbine combustor design methodologies have evolved from "trial and error" approaches to include semi-empirical, analytical and experimental evaluation approaches [1]. Recently multidisciplinary design optimization (MDO) techniques have also begun to be applied to automatically improve a combustor design [2, 3]. While such techniques succeed in automating the design process to some extent, thereby reducing both development costs and time-to-market, some human interaction in the creation of such optimization workflows is unavoidable [4, 5] and can be quite time consuming when developing a workflow to cope with complex topology changes.

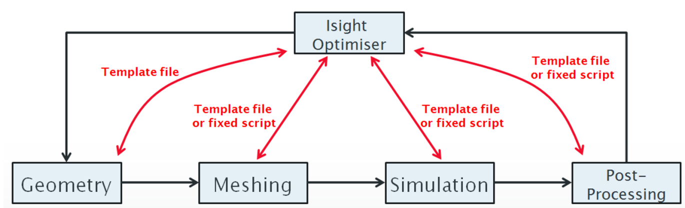


FIGURE 1. A TRADITIONAL OPTIMIZATION WORKFLOW

A typical traditional "optimizer-centric" optimization workflow, illustrated in Fig. 1, usually employs the optimization program to control each component and manage the generation of any necessary script files [6]. For simple design problems such an approach may be adequate, however, this "optimizer-centric" approach limits the designer's ability to consider topological changes which may, in some cases, invalidate the remaining steps in the workflow.

The optimization workflow of Kenworthy and Jensen [6], for example, requires the user to manually provide input and output files for various workflow components. The manual redefinition of each script is therefore required when considering a different topology or, alternatively, a complex system to auto-

[*]Corresponding author: Xu.Zhang@soton.ac.uk

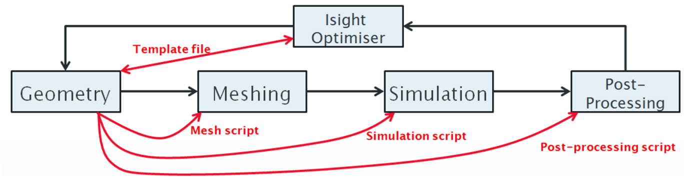


FIGURE 2. PROMETHEUS' "GEOMETRY-CENTRIC" OPTIMIZATION WORKFLOW

matically change the script type must be codified within the optimizer. However, to cope with such changes such a system would require additional, more complex, links between the optimizer and the geometry thereby considerably increasing the complexity of the optimization workflow.

Examples of this issue appear numerous times throughout the design optimization literature. The combustor design optimization of Duchaine et al. [3], for example, uses a series of fixed planes to post-process CFD simulations in order to calculate the objective functions. A significant topological change to the combustor may therefore render these post-processing planes invalid. To consider a new combustor geometry the designer would therefore have to adjust the definition of these planes.

Rogero and Rubini [7] and Despierre et al. [8] both performed combustor optimizations using network based approaches focused on the control of flow into and out of the combustor by adjusting the dilution port geometry. In both cases, however, the network remains completely fixed, topological changes to, for example, the number of ports cannot therefore be performed as it would require the manual creation of a completely new network. Within a traditional optimization workflow, adding and removing dilution ports and maintaining the connectivity of the network clearly becomes a complex and onerous task.

Moving beyond gas turbine combustors, Brujic [4] used a parametrized CAD model for disc shape optimization. But the ability to consider significant topological geometry changes to the CAD model is still limited. For example, it is difficult to use the same workflow to add an additional arm to the disc or change the seal topology. Similarly switching between different types of blade root topologies [9] may require considerable alterations to the meshing and contact analysis scripts and changing blade tip topologies [10, 11] and endwall contouring or treatment topologies [12] may require significant changes to the CFD mesh to accurately capture the flowfield.

The following paper presents the Prometheus combustor design system which shifts the control of script generation from the optimization software to the CAD package. Unlike a traditional "optimizer-centric" optimization workflow (see Fig. 1), Prometheus aims to reduce both the level of workflow complexity and rework by taking a more "geometry centric" approach to optimization, as seen in Fig. 2. In Prometheus, it is the CAD package rather than the optimizer which manages the generation of script files for each component in the workflow thereby allowing geometry changes to be automatically reflected in any generated scripts. By adopting such a geometry centric approach there is now only one input file controlling the whole workflow. If any change has been made to the geometry, Prometheus will automatically reflect these changes in all necessary scripts. Significant topological geometry changes within the design study can therefore be applied much more easily which simultaneously reduces workflow complexity while increasing the level of geometric flexibility available to the designer.

To illustrate this further let us consider the case previously mentioned where a designer wishes to include an additional row of dilution ports in the combustor. Alterations to the geometry can be performed relatively easily but the design requires that new mesh refinement zones be added to accurately capture the flow through the new ports. This information is absent in the scripts from a previous design study and it may require considerable effort to update the script files either manually or through code redevelopment. Prometheus, however, uses a series of feature based geometry recognition routines to allow geometry changes to be automatically reflected in any generated scripts for a variety of operations. Prometheus also therefore permits engineering knowledge and best practice to be codified thus ensuring that all designers are using common approved approaches. This embedded knowledge can, of course, be updated or appended as guidelines mature.

This paper is organized as follows. Section 2 provides an overview of the Prometheus system. Then, the fluid volume generation, meshing, aero-thermal network generation, CFD and post-processing are detailed in Sections 3-6 respectively. Section 7 demonstrates the ability of Prometheus to create the fluid volume, network analysis and scripts after a combustor has undergone a significant geometry change.

2 OVERVIEW OF PROMETHEUS

The main goal of Prometheus is to develop an efficient and effective multi-disciplinary combustor design and optimization system. As seen in Fig. 3, Prometheus is developed using the standard C++ object-oriented programming approach and a mixture of the Siemens NX Open C and C++ application programming interfaces (API). Special emphasis is given to the application of the NX Open API to efficiently automate various stages of the optimization loop, including geometry generation, modification, identification, aero-thermal network generation, and mesh and CFD preparation. There are two parallelly-developed versions of Prometheus. The dynamic linked library (.dll) can be used interactively within Siemens NX while the executable version (.exe) can be run in batch mode within an optimization. To manipulate the combustor geometry and efficiently perform an optimization, Prometheus enables the user to perform a variety

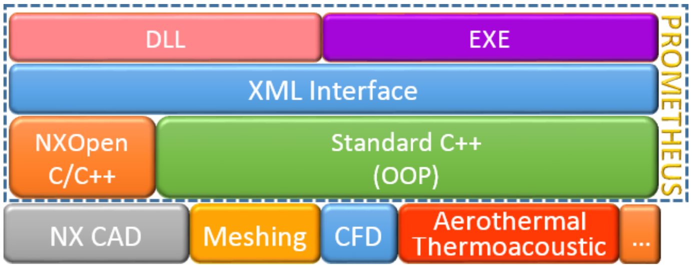


FIGURE 3. PROMETHEUS PLATFORM HIERARCHY

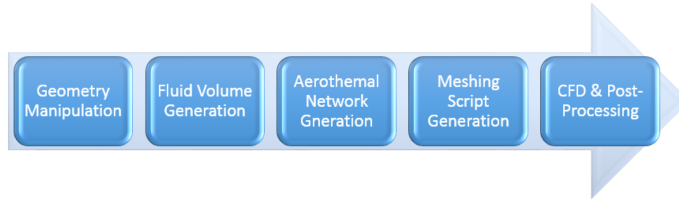


FIGURE 4. THE GENERAL PROMETHEUS PROCESS

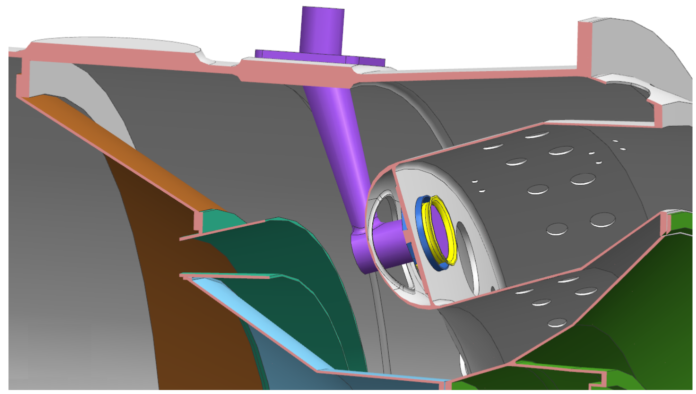


FIGURE 5. AN EXAMPLE COMBUSTOR GEOMETRY

of operations via a simple eXtensible Markup Language (XML) interface with pre-defined operation tags.

Apart from using C++ and the NX Open API, Prometheus aims to utilize the existing software tools and best practice within the Rolls-Royce combustor design team. For instance, it can link directly to any combustor preliminary design tools proceeding it within the design cycle and use information from these tools to help define geometry and simulation boundary conditions. To analyze the aero-thermal performance of an individual design, an aero-thermal network can be generated automatically based on the generated fluid volume. Moreover, the obtained network also contains necessary information for thermo-acoustic performance analysis carried out using the Low-Order Thermo-Acoustic Network (LOTAN) solver [13–15]. The ANSYS ICEM CFD mesher can be used to import the fluid volume geometry and create a mesh by running the automatically generated meshing script. Similarly a CFD simulation of the combustor design can be performed using the Predictive-System for Real Engine Combustors (PRECISE) using the automatically generated script and associated mesh. The longer term goal is to extend Prometheus' ability to include structural analysis, through SC03 [16, 17], and cost-modeling [18].

An illustration of the general Prometheus process is presented in Fig. 4. Using Prometheus' direct links to Siemens NX, the combustor geometry can be manipulated using existing parametric features or through NX's historyless modeling capabilities. With geometry modifications completed an appropriate fluid volume is generated which is itself used to define an aero-thermal network and a corresponding ICEM CFD meshing script. The results from the completed aero-thermal network simulation are then used to automatically define boundary conditions for the CFD simulation.

In the following sections, each of these individual operations will be discussed in detail with respect to the combustor geometry illustrated in Fig. 5.

3 FLUID VOLUME GENERATION

Traditionally the creation of a CFD fluid volume requires an experienced engineer to explicitly define the fluid domain using the CAD software's graphical user interface (GUI). While some CAD packages have a number of inbuilt tools to aid in the creation of such volumes these rely on the specification of global parameters which can result in regions necessary for an accurate CFD simulation being excluded and others not necessary being included. Although these tools are explicitly linked to the geometry within the CAD history tree, significant topological modifications made via a historyless approach may cause these links to break down thereby invalidating the fluid domain. Similarly this approach of creating a fluid volume contains no intelligence regarding the features it's extracting resulting in a "dumb" fluid volume with explicit links to any simulations missing. An approach is therefore required which has the intelligence to include the desired level of geometric fidelity within the fluid volume, is robust to design changes and can recognize features important for the definition of the simulation.

The fluid volume generation within Prometheus is based upon a feature recognition process which is used to define predetermined useful or desirable features for the fluid volume to contain. This not only helps to define which solid faces should be used to represent the surfaces of the fluid volume but also any important features necessary for the application of boundary conditions and mesh refinement zones.

Figure 6 illustrates how these feature identification routines are built up, in a hierarchical manner, from the initial building blocks of the native NX Open API functions. These native functions perform very simple operations on the CAD geometry, they

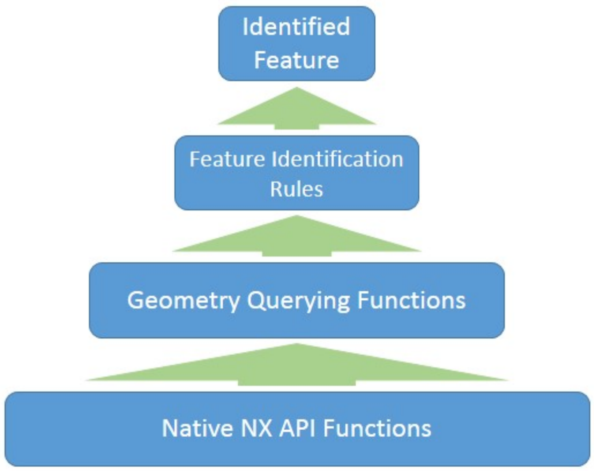


FIGURE 6. FLUID VOLUME FEATURE RECOGNITION FUNCTION HIERARCHY.

can, for example, extract the faces of a solid, calculate the distances between two entities, create curves, create extrusions etc. These basic API functions are then compiled into a series of approximately 14 higher level geometry querying functions. For each feature in the combustor a set of boolean identification rules are then defined using these querying functions. When these rules are applied to a particular geometric entity then the required feature is identified.

To illustrate the application of these rules let us consider the example of the combustor outer casing illustrated in Fig. 7. In order to construct the fluid volume it is necessary to identify faces on the inside of this casing. After partitioning the casing into a single section, whose size corresponds to the number of injectors in the combustor, the feature identification rules are applied. Firstly a set of rules is defined to identify the rear flange face, then a similar set of rules is used to identify the foremost flange face. The fluid volume is therefore defined by the surfaces between these two end faces (highlighted in red). The extraction of faces to the fluid volume therefore begins at the rear flange and moves toward the front flange ignoring any face which lies wholly on a periodic boundary (highlighted in green). With these three simple operations the faces required from the outer combustor casing can be identified and used to create the fluid volume.

A similar process is used to identify the remaining faces of the fluid volume and is analogous to rolling a ball within the combustor and extracting only those faces which it touches.

With the main faces identified the same geometry querying functions can be used to identify more intricate features. Take the lower flametube casing illustrated in Fig. 8 as an example. Here the main faces of the casing necessary for definition of the fluid volume have been identified. The face highlighted in red

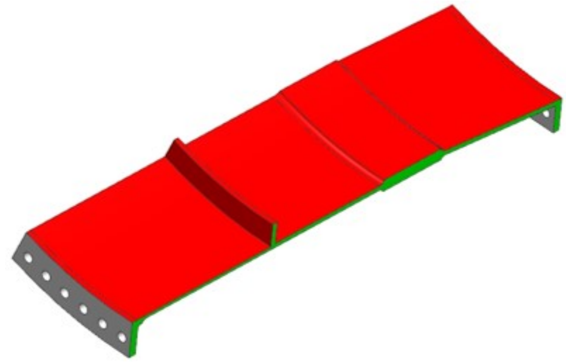


FIGURE 7. OUTER CASING WITH FLUID DOMAIN FACES IDENTIFIED.

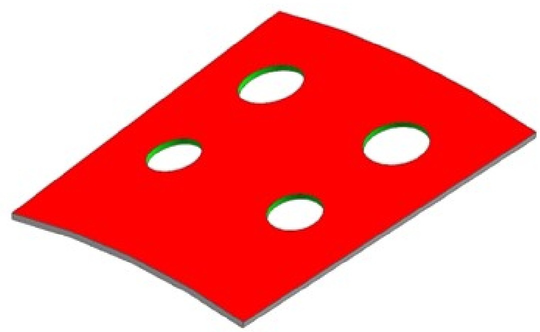


FIGURE 8. FLAMETUBE SECTION WITH PORTS IDENTIFIED.

corresponds to the flametube and the opposite face is part of the combustor annulus. However, this casing also has a number of dilution ports which must be included in the final fluid domain. A rule is therefore defined to recognize those faces making up a dilution port based on the shape of the port and the connectivity of its defining faces between the flametube and the annulus. Using this simple rule all of the dilution ports in the combustor can be identified whether they are simple holes (as illustrated in Fig. 8) or more complex chuted ports.

The identification of such features not only ensures there are no leaks in the surfaces defining the fluid volume but also enables the creation of the aero-thermal network and CFD mesh. Identifying each port in this manner enables the diameter, pitch, plunge radius and axial position to be automatically calculated. By then grouping similar dilution ports together the number, type and position of the ports around the complete annulus can be used during the aero-thermal network generation. Similarly, given the importance in resolving the flow entering the flametube through such features, mesh refinement zones can be automatically placed around every port where required.

A similar process is also used to identify vanes within any air swirlers and then based on the swirler passage size an appropriate mesh refinement zone can specified and indeed an appropriate passage volume can be calculated for any subsequent LOTAN simulation.

Also included within the fluid volume generation process is the creation of surface splits and additional surfaces for the application of CFD boundary conditions. The original 3D geometry, for example, does not contain inlet or exit planes along which to prescribe boundary conditions, nor does the original geometry contain the required surface splits to define, for example, effusion boundary conditions. These are all defined during the fluid volume creation process and, where necessary, linked to any preliminary design tools defining, for example, the number of effusion patches.

The combustor inlet planes are created by first automatically identifying the upper and lower faces of the diffuser. The foremost edges of both of these faces are then automatically identified and the intersection of each of these edges with the XZ plane is calculated. These two points form the start and end of a spline defining the inlet plane. Also helping to define this spline is the normal direction of both the inner and outer faces of the diffuser at these points. Constructing the spline through these two points to match the normals and revolving it through the sector results in an inlet plane which is normal to both surfaces of the diffuser. Retaining this normal direction allows the velocities to be defined tangent to both diffuser surfaces. The construction of the flametube exit is a similar process but also includes the creation of a representation of the NGV platform who's dimensions can be controlled via the XML input file if desired. The outer annulus exit boundary is constructed by projecting the edges of the sheets defining the rear mounting struts onto the XZ plane. These projections are then combined into a single curve which is rotated around the combustor axis. This results in a single sheet spanning the combustor sector which can be trimmed using the mounts to produce the exit planes.

Once defined the properties of these boundary conditions can be extracted from the preliminary design tools or explicitly set up using the XML input file and used to define the appropriate simulation input file. Figure 9 illustrates the fluid volume automatically generated from the combustor geometry illustrated in Fig. 5.

In addition to the creation of boundary condition application zones, surfaces are also automatically created for the purposes of simulation post-processing. In this case surfaces are created in both the inner and outer annuli, inside each of the air swirlers and close to the combustor exit. The relative position of each of these planes is fully parametric which allows, for example, the post-processing plane close to the combustor exit to be moved to a location where experimental data is available. While the annuli and swirler planes can also be used for post-processing, their main purpose is to help define the boundary conditions for a reduced annuli only model. In such a case the flowfield conditions along these boundaries can be extracted from the complete combustor simulation and used in a cheaper annuli only model. The existence of two levels of CFD fidelity coupled with a simple means to control mesh size permits the future application of multi-fidelity methods [1, 2, 19] within an optimization. Figure 10 illustrates the creation of a flametube-and-annulus fluid volume from a complete fluid volume via a simple truncation of the model controlled through a single XML command.

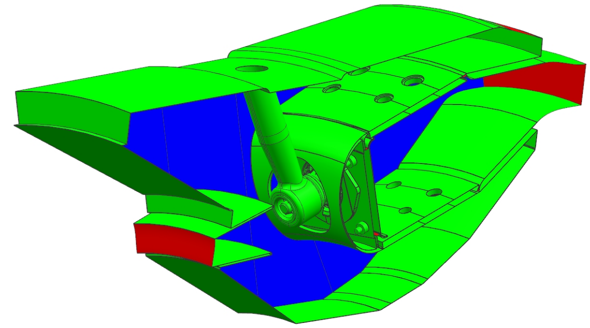


FIGURE 9. AUTOMATICALLY CREATED COMBUSTOR FLUID VOLUME.

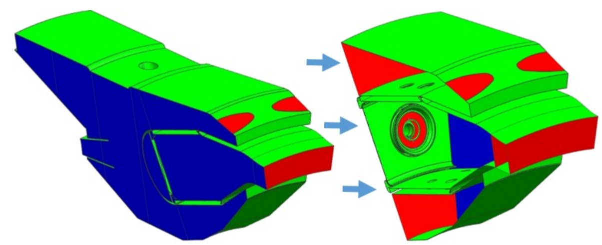


FIGURE 10. SIMPLIFIED FLAMETUBE-AND-ANNULUS FLUID VOLUME.

At this point it should be clarified that it is not the role of Prometheus to automatically decide upon which geometric changes to make. Rather it is the role of Prometheus to alter the geometry according to a CAD parameterization previously defined by the engineer and pre-process the geometry for analysis. The overall optimization process therefore still requires the engineer to make a decision regarding the features to be altered and appropriate limits for the magnitude of these changes.

4 FLOWNET NETWORK GENERATION

Prometheus is capable of performing an assessment of the aero-thermal performance of a combustor design by automatically generating and solving a Flownet network. Flownet is a network-based flow solver used to model gas turbine combustion

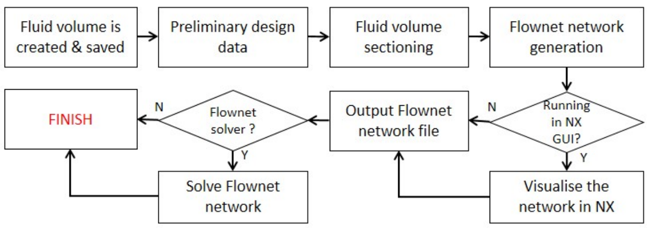


FIGURE 11. FLOWNET NETWORK GENERATION PROCESS

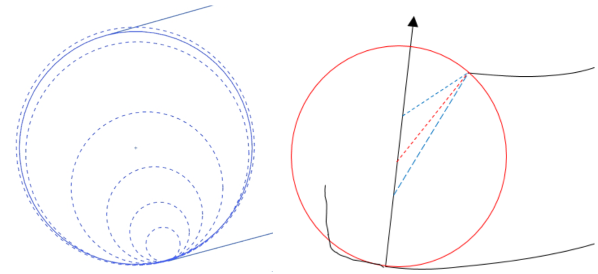
chambers and reheat systems and is used as a preliminary design tool, to provide boundary conditions for CFD and thermal analyzes and generate LOTAN models for calculating combustion instabilities. For more details on Flownet and its applications please refer to the literature [7, 8, 20]. The general Flownet network generation process is illustrated graphically in Fig. 11.

With the fluid volume generated, Prometheus commences the network generation process by extracting any necessary data from the Rolls-Royce combustor preliminary design tool. Defined within an Excel spreadsheet, operating conditions, cycle information and information on parts of the geometry not explicitly modeled via CAD are all extracted by Prometheus and used to create the network. As Excel files are essentially reformatted XML files, existing functions defined within Prometheus to parse the XML file controlling Prometheus' operations are redeployed to extract all of the required data from the preliminary design spreadsheet.

The structure of the Flownet network is actually a 1-D representation of the topology of the fluid volume geometry. To place the elements at appropriate positions in order to represent the flow features along the pre-diffuser, annuli and flametube, the centerlines of these regions must be extracted. Within Prometheus a rolling ball algorithm is used to calculate these centerlines.

The rolling ball algorithm for finding the mid-boundary or centerline between two curves is based on the fact that the centerline should be a curve running from center to center of the largest ball possible rolling between the two boundaries. Figure 12(a) illustrates the general rolling ball process. By starting from a small enough radius, the ball is grown in a direction perpendicular to an initial curve until an intersection between the circle and the second curve is found. The centroid is then saved, the circle rolls to the next point where this is repeated until the ball reaches the end of the curve. Finally, the centerline of the two curves can be defined by connecting all of the saved points into a smooth spline.

Rather than repeatedly draw circles and test their intersections Prometheus employs a more efficient approach. As the circle of interest is centered along a direction perpendicular to the



(a) (b)

FIGURE 12. ROLLING BALL (a) & EQUIDISTANT POINT SEARCH (b)

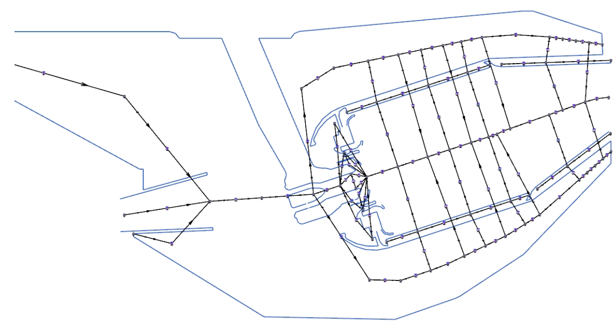


FIGURE 13. AUTOMATICALLY CREATED FLOWNET NETWORK.

tangent of the curve a binary search process can be used along this direction to find the specific point equidistant to both curves. With the dimensions of the circle defined a final check on the nature of the intersection of the circle can be made to ensure, for example, there are no secondary intersections with the initial curve. Figure 12(b) illustrates the binary search process on the diameter of the circle and also the rejection of the defined circle based on a secondary intersection with the initial curve.

With the centerlines defined the construction of the network topology can commence. The pre-diffuser centerline is first used to define the positions of a number of duct elements as well as inlet and outlet areas. The network around the swirler is then defined with paths used to represent swirler passages and leakage flows around the swirler and between the heat shield. Also calculated at this stage is a representative swirler length for use in any subsequent LOTAN simulation.

Appropriate network components representing both the ports and effusion patches are then included within the network with their properties defined using information extracted from both the CAD model and any preliminary design data. Ducts are then placed along both the annuli and the flametube to connect the network together. Once completed, checks are performed on

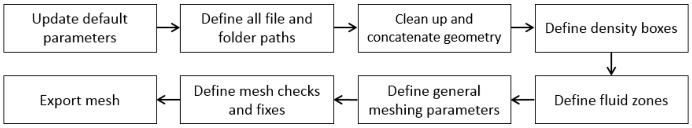
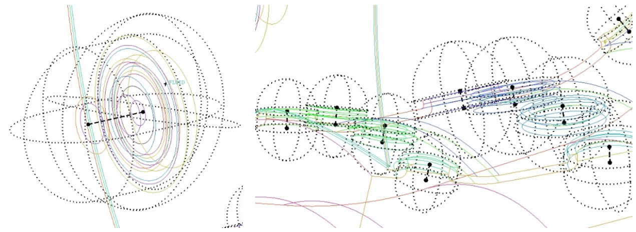


FIGURE 14. ICEM CFD MESH SCRIPT GENERATION PROCESS



(a) (b)

FIGURE 15. ICEM MESH REFINEMENT ZONES AUTOMATICALLY PLACED AROUND THE SWIRLER (a) AND DILUTION PORTS (b)

the connectivity of the final network and to ensure an appropriate number of elements in critical regions. Figure 13 illustrates the Flownet network automatically created from the combustor geometry of Fig. 5.

With the network created a Flownet network file is written out and the network simulation automatically performed. Prometheus will then parse the pressures, temperatures, mass flows, air fuel ratios and heat transfer coefficients of the solved network and store this information within memory. An NX part file containing a geometric representation of the network topology and the results of the network simulation will also be created and saved for further study. If required the results of the network analysis can then be used to define the boundary conditions for a CFD simulation of the combustor.

The modular structure of the network generation process allows Prometheus to include additional, external, network templates during the generation. This permits existing manually defined networks for certain components to be reused within the automated process. A predefined network for a new type of injector, could, for example, be read in by Prometheus and joined up to the remaining network.

5 MESHING

As previously described, during the fluid volume generation process Prometheus will intelligently identify important features within the combustor design. As well as being used to define the aero-thermal network this information is also used to set up an appropriate meshing strategy for the fluid volume. In this case a script for ICEM CFD is automatically written out.

Figure 14 outlines the process for generating such a meshing

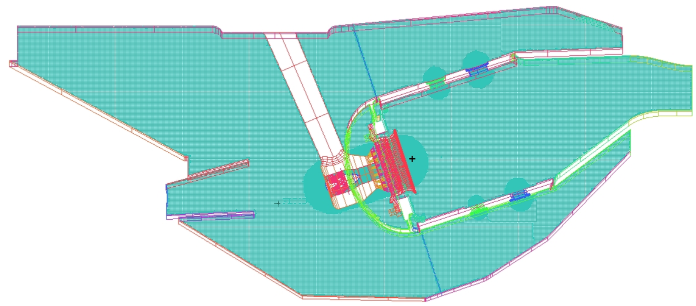


FIGURE 16. MESH GENERATED USING AN AUTOMATICALLY CREATED ICEM MESHING SCRIPT.

script. Firstly, Prometheus updates the embedded default parameters to those provided by user through the XML input file. After processing all essential file paths, Prometheus then defines the ICEM operations to import the NX fluid volume part file and concatenate separate geometric entities into more manageable groups with the separate parts defining every surface combined into a single entity. The commands to define any mesh refinement zones in appropriate regions of the fluid volume, such as around the swirler and ports, as shown in Fig. 15, are then written to the script file. The required mesh sizes in these regions are calculated to ensure that the minimum acceptable number of cells across the narrowest flow passage is achieved. Given this mesh size and a target global mesh size, which has either been calculated based on the combustor exit height or explicitly provided by the user, Prometheus will define as many refinement zones as necessary to reach the target global mesh size as closely as possible.

Scripting commands to define fluid points, the general meshing parameters including the angle of periodicity, thin cut pairs, tetrahedral mesh generation, smoothing and hex core conversion are then written out. Finally Prometheus will write out commands to check for and fix problems with the mesh and to export the mesh to a format appropriate for PRECISE. Figure 16 illustrates the final mesh resulting from a Prometheus generated script for the geometry presented in Fig. 5 while Fig. 17 illustrates a close up of the refinement zones around the dilution ports.

As briefly mentioned above Prometheus also enables most meshing parameters to be adjusted by the user through its XML interface. Parameters such as global mesh size, the minimal aspect ratio to convert from an unstructured tetrahedron mesh to a hexahedron mesh and the minimal number of cells across a refinement zone can all be altered. This gives the user the ability to perform mesh dependence studies at will, while a varying mesh density permits the use of multi-fidelity optimization approaches.

While Prometheus' mesh generation process is currently geometry driven the creation of mesh refinement zones around the above features is a result of significant previous experience in the modeling of combustion systems. While perhaps not appli-

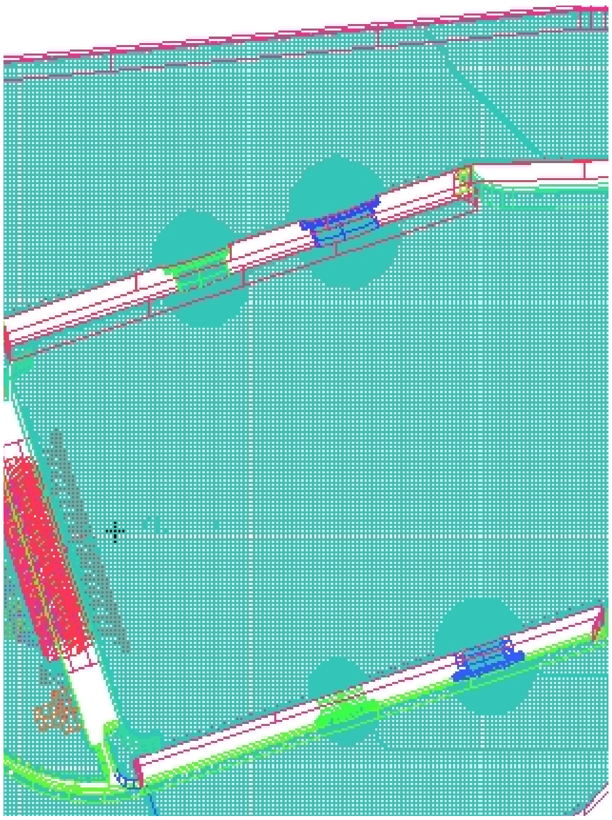


FIGURE 17. A CLOSE-UP OF THE DILUTION PORT MESH REFINEMENT ZONES.

cable to all cases, further rules to include an appropriate mesh in regions prone to separation and to control the size of the near wall mesh could be included in the future using a similar purely knowledge based process or with the help of results from the network analysis.

6 CFD SIMULATION

Given a successful Flownet network simulation, Prometheus will use the results, together with parameters calculated from the geometry and other default or user defined parameters, to automatically create a PRECISE input file. The general process which Prometheus follows to create such a file is outlined in Fig. 18.

The generation of the PRECISE input file commences with the definition of the solver parameters, the physical parameters and the model parameters. Each of these values is set as a particular default value but can also be adjusted manually via Prometheus' XML input file. Prometheus will then include the commands to define the fuel injection location within the input file. The position is calculated based on the combustor geometry

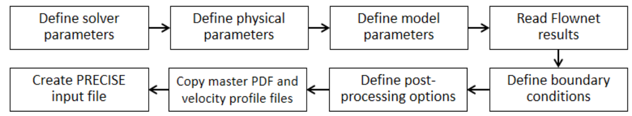


FIGURE 18. PRECISE SCRIPT GENERATION PROCESS

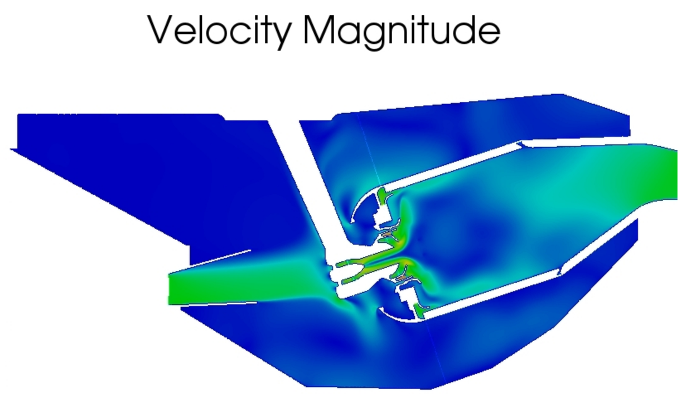


FIGURE 19. CONTOURS OF VELOCITY MAGNITUDE ON THE XZ PLANE FROM A CFD SIMULATION DEFINED BY PROMETHEUS

although it can be over-ridden by the user via XML. Prometheus will then write out the commands to define any remaining boundary conditions for various inlets and outlets within the model.

With any fuel spray nozzle defined Prometheus will then write out the commands to define any remaining boundary conditions for various inlets and outlets within the model. While velocities, pressures and mass flows can be automatically extracted from the Flownet simulation and used to define these boundary conditions the user is also free to specify predefined velocity profiles at these boundaries. Finally, Prometheus will define the commands to post-process NOx, soot and other quantities of interest from the CFD simulation over the planes created during the fluid volume generation.

Figure 19 illustrates the results of a PRECISE simulation performed using a Prometheus generated mesh and input file where the mass flow splits for the combustor outlets as well as the effusion and secondary flow inlet mass flows have been automatically defined based on the results from an automatically generated Flownet simulation. The main inlet mass flow to the combustor is also taken from the Flownet simulation but this quantity is itself defined by the combustor preliminary design tool. While Prometheus is capable of applying a predefined velocity profile to the combustor inlet, in this example the mass flow is defined along the x-axis and is uniform over the inlet. It should be noted that the inputs from the preliminary design tool to the Prometheus system are, in this instance, not wholly representative of the combustor geometry being simulated. The CFD results presented in Fig. 19 are therefore purely an illustration of the final result of the Prometheus process.

Never-the-less with the successful completion of this CFD simulation Prometheus has therefore succeeded in moving the generation of all scripts from the optimization program and into the CAD package where native geometry querying functions can be readily exploited and where engineering best practice can be easily stored.

7 EXAMPLE COMBUSTOR MODIFICATIONS

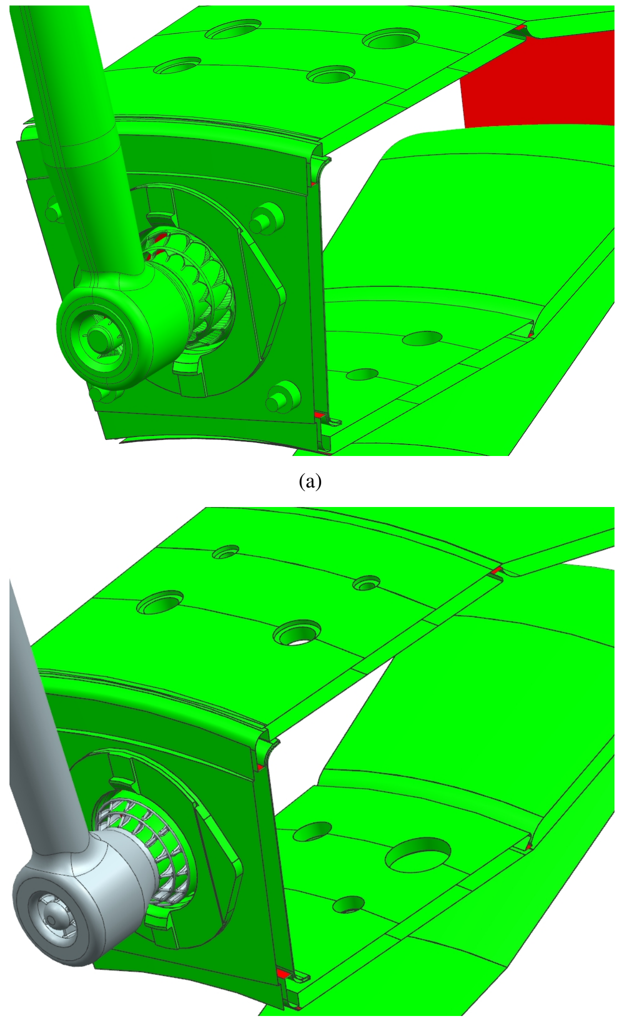
Having fully described the capabilities of the Prometheus combustor design tool with respect to automatically defining Flownet networks, CFD fluid volumes and meshes let us demonstrate these capabilities further through a series of example modifications to the combustor design illustrated in Fig. 5.

To recap from the previous sections, Fig. 9 illustrates the fluid volume generated from this baseline combustor geometry while Fig. 13 illustrates the corresponding Flownet network. Figure 16 illustrates the resulting mesh and Fig. 17 illustrates the mesh refinement zones around the dilution ports.

The original combustor design illustrated in Fig. 5 has a total of eight dilution ports, four along both the inner and outer annuli with the initial row of ports on both annuli slightly smaller than the rear row. The original design also includes a representation of both the nuts and bolts holding the heatshield in place and an intricate air swirler geometry.

Consider now the automatic generation of the fluid volume, aero-thermal network and mesh for a modified version of this combustor geometry. In this case the second row of ports on both the outer and inner annuli have been modified. The outer ports have all been reduced in diameter while a single port per sector on the inner secondary row has been increased in diameter. While not perhaps a typical design change this makes it easy to illustrate an automatic change in the topology of the aero-thermal network. The representations of the heatshield nuts and bolts have also been removed to simplify the simulation and the intricate air swirler geometry present in the original combustor has been replaced with a different design.

These modifications have been achieved through a mixture of modifications to the XML input file and manual modifications to the CAD geometry. To swap the swirler for a different design the XML has been modified to point Prometheus to the new swirler CAD part. During the fluid volume creation the heatshield pin features are automatically identified using the feature recognition rules. The size and position of these pins are then used to define the position of the pin and nut in the original fluid volume, Fig 20(a). A single XML command can therefore be



(a) (b)

FIGURE 20. CLOSEUP OF AIR SWIRLER IN BASELINE (a) & MODIFIED GEOMETRY (b).

modified to adjust the fidelity of this representation. In this case the command is set so that no pins are included within the geometry at all. As the original geometry has no capability to produce staggered port diameters, Siemens NX's in built historyless modeling tool is used to alter the ports. Whilst this is initially a manual change the historyless tool permits these new diameters to be set as expressions within the model. Through its XML interface Prometheus is fully capable of modifying any expressions within a model and now that the ports have been parameterized they can be easily altered via the XML within a future design loop.

Given these example geometry modifications the Prometheus system will automatically create the appropriate fluid volume, aero-thermal network and meshing script. Figure 20 illustrates a close-up of the region around the flame-

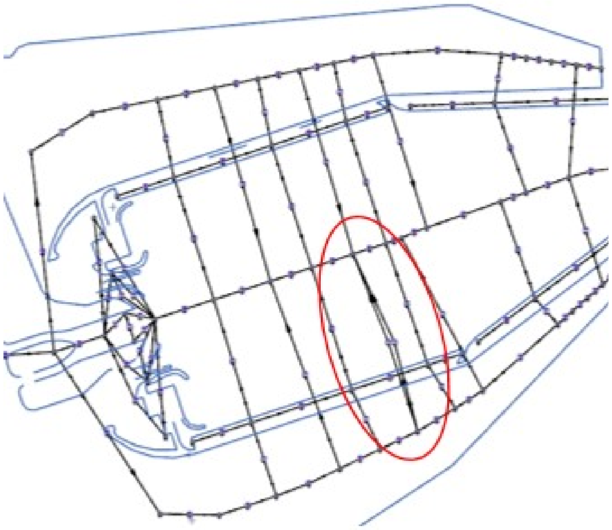


FIGURE 21. CLOSE-UP OF THE FLOWNET NETWORK RESULTING FROM MODIFIED GEOMETRY.

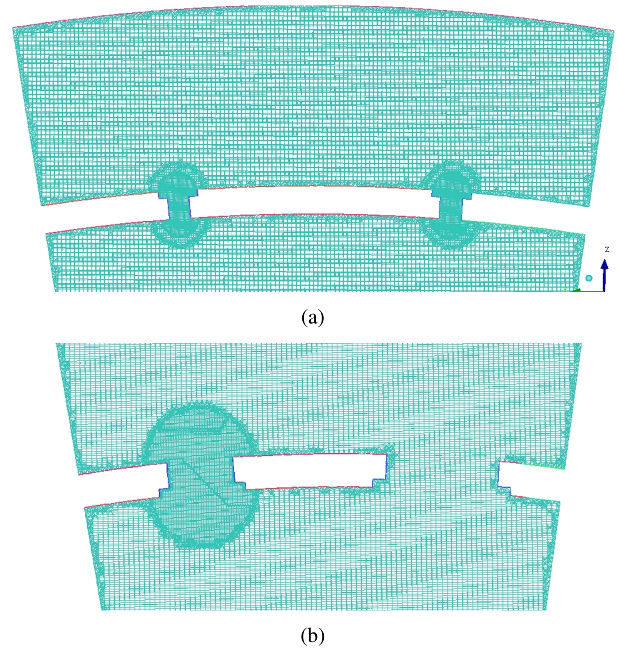


FIGURE 22. MESH CROSS-SECTIONS THROUGH THE OUTER (a) & INNER (b) SECONDARY PORTS OF THE MODIFIED COMBUSTOR.

tube of both the original and modified geometries. Note that in this figure the cowling of the combustor has been hidden so that the changes are easier to observe. Figure 20(b) clearly illustrates the changes made in the design of the dilution ports. The reduction in the diameter of the outer secondary row of ports has clearly been reflected in the fluid volume as has the change in the diameter of the single port on the inner annulus.

Also clearly illustrated in Fig. 20(b) is the removal of the heatshield nuts and bolts and the switch in air swirler geometry. Comparing Figures 20(a) and 20(b) the differences in the geometry of the two swirlers can be clearly observed. The original swirler is considerably more complex, the stem connecting the swirler to the casing has a varying cross-section whereas it's a simple tapered cross-section in the new design. The region around the inlet to the innermost swirler is also very different. The original design has a flat inner hub whereas the new design is rounded. The vanes in the new design are also substantially different to the original. Blends are now present between the vanes and the faces of the swirler passage and the vane is represented by a single B-spline surface rather than a faceted surface. The entrances to the two outermost swirler passages are also much simplified over that of the original design.

The aero-thermal network illustrated in Fig. 21 has been automatically generated from the fluid volume in Fig. 20(b). It should be noted that those parts of the network not illustrated in Fig. 21 are topologically the same as the complete network illustrated in Fig. 13. For clarity Fig. 21 highlights the network within the flametube which is modified as a result of the above geometry changes.

Figure 21 illustrates that Prometheus has identified that the inner secondary row of ports has been altered. As there is no longer a single port "type", that is a set of ports with common dimensions, within the secondary row of ports, modeling this row using a single port "element" within the network is incorrect. Prometheus has automatically recognized this from the geometry and modified the topology of the network appropriately. In Fig. 21 an additional element within the circled region has been included within the network to represent the presence of two types of port within a single row.

The automatic identification of the dilution port's size and position is also reflected in the computational mesh within the flametube, illustrated in Fig. 22. As per the original geometry Prometheus has automatically included mesh refinement zones where necessary. Figure 22(a) illustrates a section through the mesh of the modified outer dilution ports. As mentioned previously, Prometheus will automatically extract important information about each port feature for use in both the Flownet generation and the placement of mesh refinement zones. In this case Prometheus has compared the diameter of the ports to the global mesh size and concluded that more than one refinement zone is required to meet the required minimum number of cells cross the port. In this instance Prometheus has not just altered the size of the mesh in the refinement zone but has also included an addi-

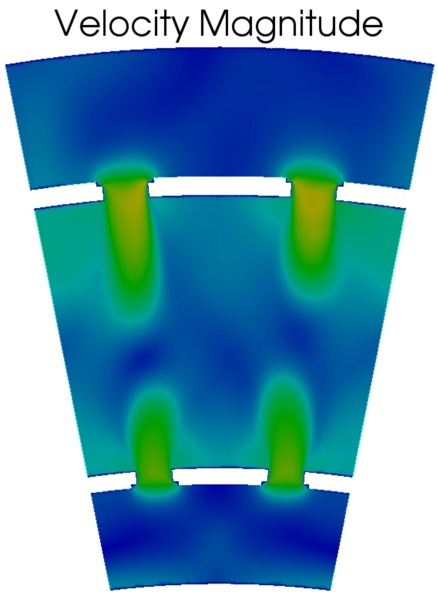


FIGURE 23. VELOCITY CONTOURS THROUGH THE SECONDARY PORTS FOR THE ORIGINAL DESIGN.

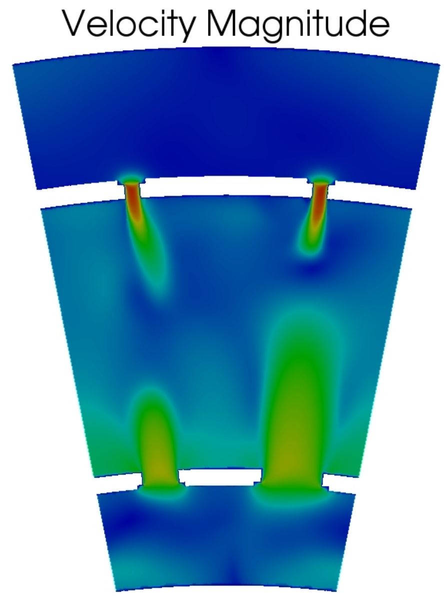


FIGURE 24. VELOCITY CONTOURS THROUGH THE SECONDARY PORTS FOR THE MODIFIED DESIGN.

tional secondary zone to improve the growth of the refined region out into the annulus and flametube.

Figure 22(b) illustrates a section through the mesh of the inner secondary row of ports. As per the outer row of ports, the modification to the combustor geometry has clearly been reflected in the mesh with the diameter of a single port in this row increasing. Unlike the outer ports the diameter of this port has increased to a point where the global mesh size is sufficient to provide the minimum number of elements across the port. When automatically creating the meshing script, Prometheus has not therefore included the commands to place a refinement zone around this port. The dilution port on the left of Fig. 22(b) has not been altered from the original geometry and does therefore require the application of a refinement zone. In this case only a single zone is necessary to give the required number of cells across the port.

Given an automatically generated Flownet simulation and computational mesh Prometheus can automatically generate an appropriate CFD simulation input file and use this to simulate the geometry. Figure 23 presents a contour plot of velocity magnitude along a plane through the center of the secondary row of ports of the geometry simulated in Fig. 19. Figure 24 however, presents a contour plot of velocity magnitude in the same plane for the modified combustor geometry. Comparing Figures 23 and 24 not only can the changes in port geometry be clearly observed but also the impact of these changes on the flowfield.

As the mass flow splits between the combustor and annuli outlets are similar in both the original and modified geometry the reduction in the diameter of the outer secondary row of ports has lead to an increase in the velocity of the air moving from the outer annulus through these ports into the flametube. Figures 25 and 26 indicate that the flow is also slightly accelerated through the outer primary row of ports in the modified design. Figures 23-26 indicate that the reduction in diameter of the outer secondary row of ports also has an impact on the flow through both the inner primary and secondary port rows while Fig. 27 indicates that there is a general increase in the velocity of the flow around the cowling and into the inner annulus to provide this additional mass flow into the flametube.

8 CONCLUSIONS

In the current paper the authors have presented the Prometheus combustor design system and illustrated its capabilities with respect to the automated generation of fluid domains, aero-thermal network analyzes and meshing scripts.

Based around a rule based feature recognition system Prometheus is able to generate a fluid volume and simultaneously identify important geometric features within a combustor CAD model. This feature specific information then enables the Prometheus system to automatically define mesh refinement zones and the topology of an aero-thermal network even if the combustor geometry is substantially altered.

The approach used in Prometheus enables engineering knowledge and best practice to be codified and stored within a framework which can be used by any combustor designer and

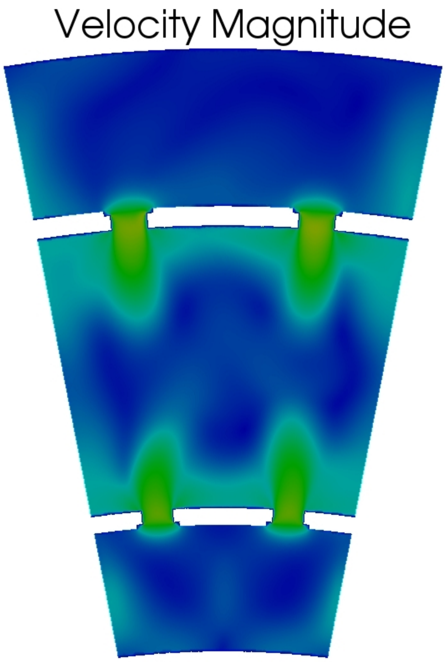


FIGURE 25. VELOCITY CONTOURS THROUGH THE PRIMARY PORTS FOR THE ORIGINAL DESIGN.

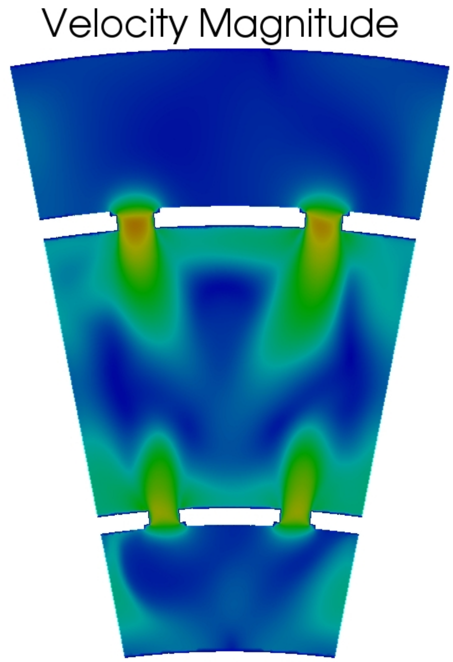


FIGURE 26. VELOCITY CONTOURS THROUGH THE PRIMARY PORTS FOR THE MODIFIED DESIGN.

easily updated to reflect the state of the art. Of course the process still requires some input from the engineer to make the geometry analyzable either by manually simplifying the CAD geometry for CFD analysis or identifying regions in the geometry to be removed or altered by Prometheus using NX's synchronous modeling capabilities.

Similarly, further work is still to be done to enhance the types of geometric features that Prometheus can cope with. Features such as outlet guide vanes and nozzle guide vanes are still to be included within the system in a manner which captures the meshing rules for these features and also permits their geometric fidelity to be altered.

While the current paper illustrates the ability of Prometheus to automatically pre-process combustor geometries for meshing, CFD simulation and network analysis the ultimate aim is to embed this system within a design optimization workflow. In such an optimization process the CFD simulation would be used to optimize the combustor for a variety of objectives and constraints including pressure recovery, exit temperature and emissions (NOx, soot etc.). Such an optimization could also be augmented with low fidelity results from a simplified annulus only model of the combustor or from the network analysis in a multi-fidelity optimization framework. Furthermore, automatic post-processing of the flowfield could be used to investigate particular regions in the combustor and automatically identify undesirable flow features. Separation in the prediffuer could, for example, be included as another constraint within an optimization.

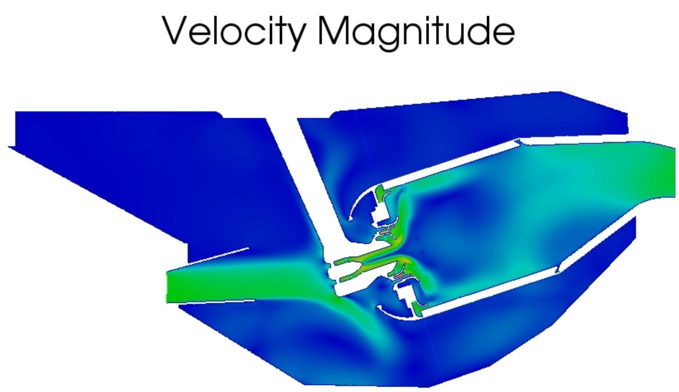


FIGURE 27. CONTOURS OF VELOCITY MAGNITUDE ON THE XZ PLANE OF THE MODIFIED DESIGN

Other future developments of the Prometheus system will aim to extend the process to include other combustor topologies, including lean burn, and the integration of structural analysis and cost modeling routines.

ACKNOWLEDGMENT

The work presented in this paper is funded by Rolls-Royce Plc. The support and collaboration from Rolls-Royce Bristol and

Derby teams are greatly acknowledged.

REFERENCES

[1] Wankhede, M. J., 2012. "Multi-fidelity strategies for lean burn combustor design". PhD Thesis, University of Southampton, Southampton, UK, January. http://eprints.soton.ac.uk/210785/.

[2] Wankhede, M., Bressloff, N., and Keane, A., 2011. "Combustor design optimisation using co-kriging of steady and unsteady turbulent combustion". In Proceedings of ASME Turbo Expo 2011.

[3] Duchaine, F., Morel, T., and M. Gicquel, L., 2009. "Computational-fluid-dynamics-based kriging optimization tool for aeronautical combustion chambers". AIAA Journal, **47**(3), pp. 631–645.

[4] Brujic, D., Ristic, M., Mattone, M., Maggiore, P., and De Poli, G. P., 2010. "CAD based shape optimization for gas turbine component design". Structural and Multidisciplinary Optimization, **41**(4), pp. 647–659.

[5] Keskin, A., Dutta, A. K., and Bestle, D., 2006. "Alternative approach for solving a multi-objective optimization problem in aerodynamic compressor blade design". Design Optimisation: Methods & Application. Proceedings of ERCOFTAC, pp. 227–230.

[6] Kenworthy, T. L., and Jensen, C. G., 2009. "CAD-centric dynamic workflow generation". Computer-Aided Design & Applications, **6**, pp. 673–683.

[7] Rogero, J., and Rubini, P., 2001. "Optimisation of combustor wall heat transfer and pollutant emissions for preliminary design using evolutionary techniques". In Proceedings of ISOABE, 15th International Symposium on Airbreathing Engines, 2-7 September, Bangalore, India.

[8] Despierre, A., Stuttaford, P. J., and Rubini, P. A., 1997. "Preliminary gas turbine combustor design using a genetic algorithm". In Int. Gas Turbine and Aeroengine Congress & Exhibition, 2-5 June, Orlando, Florida. ASME 97-GT-72.

[9] Song, W., Keane, A., Rees, J., Bhaskar, A., and Bagnall, S., 2002. "Turbine blade fir-tree root design optimisation using intelligent cad and finite element analysis". Computers and Structures, **80**, pp. 1853–1867.

[10] Kavurmacioglu, L., Dey, D., and Camci, C., 2002. "Aerodynamic character of partial squealer tip arrangements in an axial flow turbine". In IGTI/ASME Turbo Expo.

[11] Stephens, J., 2009. "Control of the tip-gap of a low pressure turbine blade in a linear cascade". PhD thesis, University of Notre Dame.

[12] Ito, Y., Watanabe, T., and Himeno, T., 2008. "Effect of endwall contouring on flow instability of transonic compressor". International Journal of Gas Turbine, Propulsion and Power Systems, **2**(1), pp. 24–29.

[13] Stow, S. R., and Dowling, A. P., 2001. "Thermoacoustic oscillations in an annular combustor". ASME Paper. GT2001-0037.

[14] Stow, S. R., and Dowling, A. P., 2004. "Low-order modelling of thermoacoustic limit cycles". ASME Paper. GT2004-54245.

[15] Stow, S. R., and Dowling, A. P., 2009. "A time-domain network model for nonlinear thermoacoustic oscillations". Journal of engineering for gas turbines and power, **131**(3).

[16] Armstrong, I., and Edmunds, T., 1989. "Fully automatic analysis in the industrial environment". In Proceedings of Second International Conference on Quality Assurance in Finite Element Analysis, NAFEMS.

[17] Edmunds, T., 1993. "Practical three dimensional adaptive analysis". In Proceedings of the Fourth International Conference on Quality Assurance and Standards, NATEMS.

[18] Wong, J., Scanlan, J., and Eres, M., 2009. "An integrated life cycle cost tool for aero-engines". In 6th International Product Lifecycle Management Conference.

[19] Kennedy, M., and O'Hagan, A., 2000. "Predicting the output from a complex computer code when fast approximations are available". Biometrika, **87**(1), pp. 1–13.

[20] Stuttaford, P. J., and Rubini, P., 1997. "Preliminary gas turbine combustor design using a network approach". Journal of Engineering for Gas Turbines and Power, **119**(3), July, pp. 546–552.