Combustor Design Optimization Using the Prometheus Design System

David J.J. Toal, Xu Zhang and Andy J. Keane
djjt@soton.ac.uk
University of Southampton
Faculty of Engineering & the Environment
Southampton
U.K.

Simon Stow and Marco Zedda
Rolls-Royce plc.
Derby
U.K.

Frederic Witham and Jonathan Gregory
Rolls-Royce plc.
Bristol
U.K.

ABSTRACT

Employing a novel feature recognition based approach, the Prometheus combustor design system enables geometry manipulation, automated aerothermal network analysis and automated fluid volume creation, meshing and CFD boundary condition definition irrespective of topological variations in the combustor configuration. To date the system has been demonstrated with respect to the isothermal optimisation of a combustor prediffuser shape and the cross-sectional shape of a fuel injector feed arm. The following paper extends this approach and presents the application of the system to the design optimisation of a single skin rich burn combustor module with reacting flow CFD simulations. A multi-objective design optimisation whereby dilution port diameters are varied to simultaneously reduce combustor pressure losses and achieve a target exit radial temperature distribution is presented. A multi-fidelity framework combining CFD and aerothermal network simulations for reducing the cost of pressure loss optimisations is also briefly explored.

Keywords: Combustor; Multi-objective; Multi-fidelity; Optimisation;

NOMENCLATURE

CFD	Computational Fluid Dynamics
NSGA-II	Non-sorting genetic algorithm II
OGV	Outlet guide vane
RMSE	Root mean square error
RTDF	Radial Temperature Distribution Function

Symbols

E_{RTDF}	RMSE in RTDF to target RTDF
ΔP	Percentage change in pressure loss relative to the baseline design
P_{Baseline}	Baseline line combustor pressure lose
P_{Loss}	Pressure loss across a combustor
$P_{T_{\text{OGV}}}$	Total pressure at the compressor OGV
$P_{T_{\text{Outlet}}}$	Total pressure at the combustor outlet
r_i	Radial location at the combustor exit
T	Combustor RTDF
T_{Target}	Target combustor RTDF
$\boldsymbol{x}$	Vector of design variables

1.0 INTRODUCTION

As with most gas turbine sub-systems, processes for the design of combustors have undergone increasing levels of automation in recent years with advanced design optimisation techniques being brought to bear in an effort to reduce emissions, pressure losses and cooling requirements [1, 2, 3, 4, 5]. Automation within such a design process inevitably reduces both development costs and time-to-market but can somewhat stifle innovation as existing workflows have to be manually rebuilt or adjusted to reflect changes in topology.

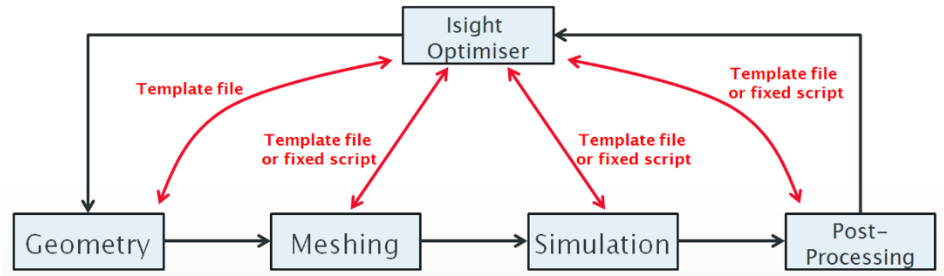

Figure 1 A typical optimiser centric workflow [6]

Typical design optimisation workflows, as illustrated in Figure 1, can be classed as optimiser centric [6]. By this we mean that geometry manipulation, pre- and post-processing and simulation activities are all controlled by a set of scripts with the optimiser or workflow environment e.g. Isight, Modefrontier etc. modifying values within these scripts before running an executable. Taking combustor optimisation as an example the workflow will modify a template expression file which defines a change to the combustor geometry e.g. a modification to the cant angle or wall profile. This geometry is then saved and the workflow proceeds to modify a template meshing file. Such modifications are generally to global settings, such as maximum/minimum cell sizes, file paths or names but could potentially include the positions of refinement zones. However, as we shall observe later, this leads to an unnecessary level of information transfer and complexity. With the meshing script defined, the mesh is generated from the geometry. A similar process is followed for the simulation and post-processing with again fairly minor changes made to template script files by the workflow.

While it is indeed possible to increase the complexity of the workflow to generate more complicated inputs to each of these routines, this increase in workflow complexity comes at the price of a reduction in workflow flexibility. With more complexity, the level of effort required to modify the workflow increases to a point whereby it could stifle innovation by preventing novel geometry configurations from being rapidly explored. While a traditional optimisation framework is extremely useful for improving a fixed topology by adjusting geometric parameters once that topology has been defined the effort in setting up a workflow to automatically consider topological variations precludes the use of automation early in the design process. This prevents the benefits of automation, in terms of reduced development costs and reduced time to market from being fully realised.

If we continue with the combustor optimisation example, while a workflow can be defined with relative ease to consider variations in something like port diameter or wall profile it becomes more difficult if topological changes such as, the number of ports, the style of injector, the mounting location etc. are the subjects of a design study. Even changes to port dimensions or locations can cause issues if the meshing script cannot be easily updated to reflect these changes. Such a workflow requires additional information to be extracted from the geometry model which needs to be interpreted by the workflow and translated into a change in the meshing process. Even with seemingly simple geometric changes workflow complexity can grow quite quickly. In order to extend the use of automated analysis an alternative approach to the creation of such workflows is therefore required.

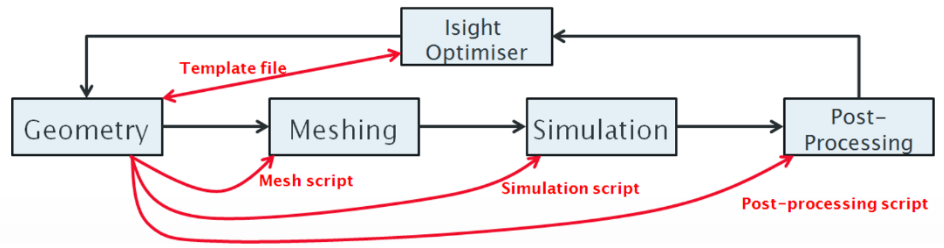

Figure 2 A representation of the Prometheus geometry centric optimisation workflow [6]

The majority of engineering design optimisation workflows commence from some definition of the geometry of either the component, sub-system or system. Irrespective of the analysis this geometry drives all analysis activities within the workflow, it drives the mesh for any structural or thermal analysis, the fluid volume and therefore the mesh for any CFD analysis. It drives the locations of joints, contacts, pressures, forces etc. within a mechanical analysis as well as the locations of inlets, outlets within an aerodynamic analysis. Even the post-processing operations are heavily driven from geometry with it defining the surfaces over which quantities of interest are integrated or displacements are measured. Given the natural tendency for geometry information to drive the definition of these processes why not use the geometry engine to directly generate such scripts within a workflow thereby removing the need to reverse engineer an understanding of the geometry within the workflow and hence removing a major cause of the overheads seen in modifying a workflow for a new topology.

The Prometheus combustor design system follows just such a “geometry centric” approach to optimisation, see Figure 2. In this system a single XML script is modified by the optimiser and an intelligent hint based feature recognition process is used to construct a CFD fluid volume of a combustor module and simultaneously identify important simulation specific features. Dilution ports, injector passages, combustor walls etc. are identified and probed to determine important dimensions, locations from which an aerothermal network can be automatically constructed, appropriate pre- and post-processing scripts and CFD simulation inputs can be defined.

The ability of the Prometheus system to cope with considerable topological changes to the geometry of a combustor has been previously demonstrated [6] as has its use within a design optimisation workflow [4]. The cited optimisation, however, involved an isothermal combustor simulation which, while undoubtedly useful when attempting to reduce pressure losses, does not demonstrate the capabilities of the system to optimise for more combustion specific parameters, such as the exit temperature profile. The following paper attempts to redress this omission by demonstrating the application of the Prometheus system to the optimisation of a single skin combustor by employing reacting flow simulations.

The article commences with an overview of the example single skin case study. The geometry is introduced as is the aerothermal network and CFD simulations, meshing process and post-processing approaches. The role of the Prometheus system within the workflow is also outlined. The multi-objective design optimisation is then introduced along with the surrogate model based optimisation strategy. Results of the optimisation are then presented and discussed. The combination of aerothermal network results and CFD simulations within a multi-fidelity surrogate modelling framework are then briefly explored as a potential method for accelerating such optimisations in the future. The paper concludes with some comments on the design system and plans for future developments.

2.0 COMBUSTOR CASE STUDY

The following section defines the combustor geometry, the simulations performed including any pre- and post-processing operations and the role of the Prometheus system within the overall analysis workflow.

2.1 Combustor Geometry Overview

While previously demonstrated on a double skin rich burn combustor [6] and a rich burn combustor isothermal rig model [4], the following case study considers the application of the system to a single skin rich burn combustor.

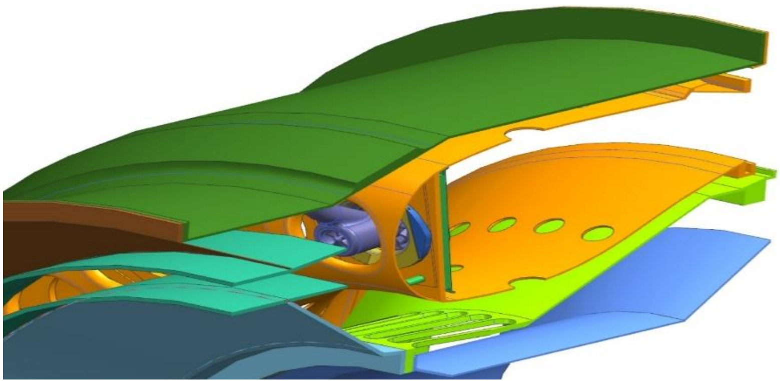

Figure 3 Single skin combustor sketchbook

The geometry itself is defined as a series of parts within the CAD package Siemens NX. The parts are linked together by a set of common parameters which define the overall shape and dimensions of the combustor within a sketchbook, see Figure 3. Separate to the sketchbook is an additional part file defining the fuel spray nozzle. In practice this part would result from a separate component design process but here is a fixed non-parametric solid body. While the sketchbook is capable of manipulating the combustor geometry in a variety of ways, for example, the number of injectors, the number of dilution ports, the cant angle, length and combustor volume, here we will focus on changes to the diameters of the inner and outer secondary rows of ports. The diameters of these ports will be permitted to vary by ±50%. It should be noted that due to the proprietary nature of the geometry all images have been rescaled and details of the internal injector passages obscured.

Given the combustor sketchbook Prometheus attempts to construct a CFD fluid volume based on its embedded engineering best practice. This process begins by converting the important faces from the original solid bodies to sheets and exporting these to a fluid volume part. The faces are identified via a hint based feature recognition process based on a combination of predefined rules which interrogate various face properties and connectivities. More information on the exact processes involved here can be found in

Zhang et al. [4, 6] for the interested reader. With the necessary sheet bodies extracted the system then constructs all inlet, outlets, post-processing planes, periodic boundaries and trims any sheets where necessary. An example of a completed fluid volume is presented in Figure 4 below.

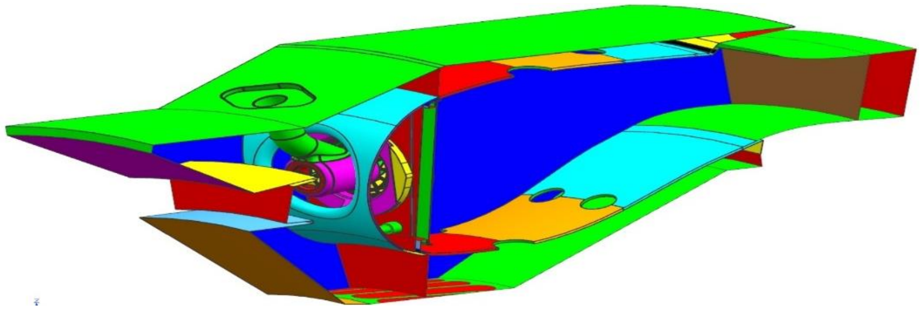

Figure 4 Prometheus generated single skin fluid volume with inlet/outlet and post-processing planes

2.2 Simulation Overview

During the feature identification and extraction process much more than just the creation of a set of sheet bodies occurs. Important features are identified and classified using the feature recognition rules which are then used to automatically construct an aerothermal network model, a meshing script, a simulation script and a series of post-processing scripts.

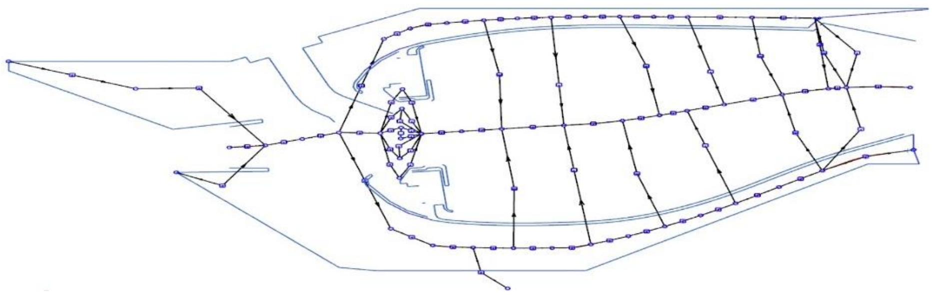

Figure 5 Automatically generated single skin aerothermal network model

The creation of an aerothermal network model, as seen in Figure 5 below, commences with both the fluid volume and associated database of identified features. From this a rolling ball algorithm is used to define the centrelines of the two annuli, flametube and the prediffuser. These centrelines are used to define duct widths within the network model and act as the main flow pathways to which flows through the injector, heatshield, effusion patches, ports etc. connect. The feature recognition of a port within the geometry, therefore is used to define the location of a port "bit" within the network and its properties, for example, its diameter, depth, plunge radius etc. Inputs to this model, for example, fuel flows, inlet mass flows etc. are then taken from the combustor preliminary design spreadsheet and the network model is solved. In this case all aerothermal network simulations are carried out using the Rolls-Royce-enhanced version of Flownet [2, 7, 8]. In addition to this information regarding the combustor design not explicitly present within the geometry are also taken from the preliminary design sheet, for example, the number size and angle of any effusion holes. Once the simulation has been successfully completed the results are read back into the CAD model and used to populate boundary conditions within the CFD simulation. Although not used here its important to note that the Prometheus system is capable of producing network models to assess thermo-acoustic performance using the proprietary LOTAN solver [9, 10, 11].

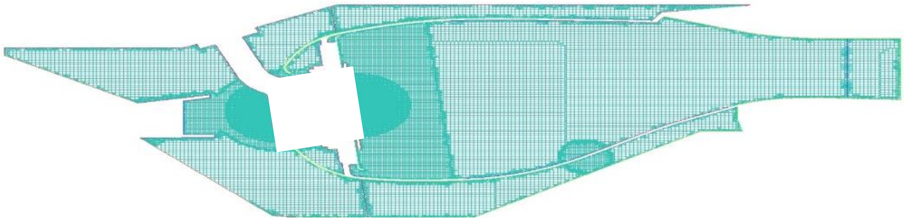

Figure 6 Mesh resulting from a Prometheus generated fluid volume and meshing script

The definition of the fluid volume part and the associated feature recognition also enables a bespoke meshing script to be automatically created. In this instance port dimensions and locations are used to define the position and sizes of mesh refinement zones to meet a defined minimum number of cells across the port. Similarly the definition of the fuel spray nozzle and the identification of its swirler passages permits refinement zones to be placed around the nozzle automatically with a cell size adjusted to meet a predefined minimum number of elements across the smallest passage gap. While each of these parameters can be controlled via the XML input to the system, default parameters stored within the program ensure that agreed meshing best practice is followed by all designers.

Figure 6 above illustrates the mesh used within the following optimisation case study. The mesh is generated in ANSYS ICEM and ranges in size between 16.9m and 17.9m elements depending on the size of the ports. A maximum global mesh size of 1.86mm is defined with a minimum of 25 cells across each port and 10 cells across the smallest swirler passage.

2.3 Simulation & Post-Processing Overview

Given a successfully generated and solved aerothermal network model, the results of this can be used to populate the CFD simulation boundary conditions. The mass flows on the effusion boundary conditions, exit mass flow splits, inlet mass flows etc. are all defined within the CFD simulation script. Other inputs such as the location and angle of the spray ring and ignitor location are extracted from the geometry. The spray ring, for example, is defined based on the fuel spray nozzle geometry and cant angle. As with the generation of the meshing script the system contains a definition of simulation best practice which is used to define, for example, the turbulence model, number of iterations, relaxation factors etc. which once again ensures consistency across all analyses.

In the following optimisations all CFD simulations are performed using the proprietary Rolls-Royce CFD solver, PRECISE-UNS [12]. The K-ε realisable turbulence model is used with simulations running initially for 12500 iterations before post-processing operations are started. Fuel spray is turned on after 500 iterations with ignition triggered 1200 iterations into the simulation.

All CFD simulations used within the present study are performed on the University of Southampton Spitfire compute cluster with each simulation run on 16 cores and taking approximately 10 days to run. It should be noted that the CFD solver is capable of running on many more cores than this but the lack of high speed interconnect currently precludes this on the Spitfire cluster. Instead each of the simulations within the study are run in parallel over a 10 day period.

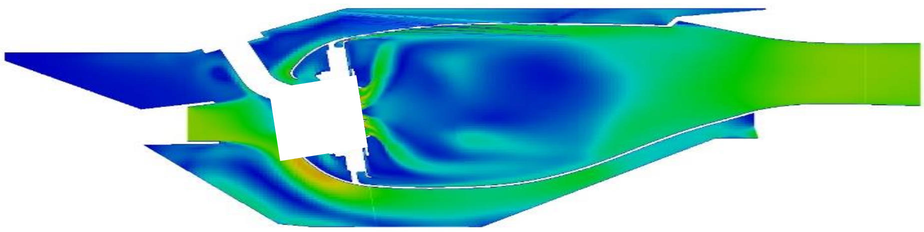

Figure 7 Contours of velocity magnitude for the baseline combustor geometry

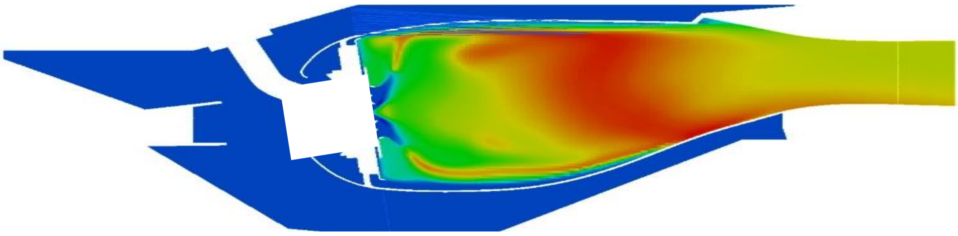

Figure 8 Contours of temperature for the baseline combustor geometry

After 12500 iterations the CFD simulation continues to run for a further 1000 iterations with results files saved in 100 iteration intervals. At each interval a VTK file is saved by PRECISE for post-processing and the RTDF at the exit of the combustor calculated. Figure 7 above illustrates contours of velocity magnitude for the baseline combustor geometry while Figure 8 illustrates contours of temperature.

As already noted the Prometheus system constructs simulations based on embedded engineering best practice and the same is true for simulation post-processing. After the creation of the simulation script a series of scripts are automatically generated by the system to evaluate various aspects of combustor performance. All scripts are generated for the proprietary Rolls-Royce variant of Paraview, SS02, and include scripts to calculate mass weighted quantities on all inlets, outlets and post-processing planes, swirl numbers, near wall temperatures, radiation and HTC maps etc. While only mass weighted pressure calculations are utilised within the presented optimisation study other quantities of interest could feasibly be used as inputs to further simulations, for example, a thermo-mechanical simulation of the combustor module.

3.0 MULTI-OBJECTIVE OPTIMISATION

Having introduced the Prometheus design system and its application within the presented case study let us now consider the optimisation processes that will be employed and the precise formulation of the design problem.

3.1 Surrogate Based Optimisation Approach

As noted above the RANS simulations employed in analysing the combustor geometry are relatively expensive. This naturally precludes their use directly within a global search algorithm such as a genetic algorithm or a particle swarm. Instead, in order to reduce the total number of simulations required, a surrogate modelling based optimisation approach is employed. Such an optimisation process replaces an expensive to evaluate simulation with models predicting the response of that simulation across a design space. These models, which can represent objective functions as well as constraints can then be exhaustively searched by a global search algorithm to locate regions of interest, for example, improved design performance. Typically the true simulation is run at this point and used to update or correct the surrogate model’s prediction. A more detailed overview of surrogate modelling, its applications and formulations can be found in the literature for the interested reader [13, 14, 15].

Within the following study an ordinary Kriging based model [16] is utilised to create surrogate models of all objective functions. This model is constructed using the propriatary Rolls-Royce optimisation suite OPTIMATv2. A hybridised particle swarm search algorithm employing an adjoint formulation of the likelihood function [17, 18] is used to construct the model and a NSGA-II algorithm [19] used to search the model.

To demonstrate the application of Prometheus within an optimisation we consider a relatively simple optimisation problem but yet one which could be considered as quite common within the combustor design community. Consider the scenario whereby the combustor illustrated in Figure 4 has been analysed and found to give a RTDF different to that agreed between the combustion and turbine sub-systems. The aim, therefore is to improve the RTDF of the combustor but not negatively impact the combustor’s predicted pressure loss. In this instance we shall permit the diameters of the outer and inner dilution ports (the second row of ports in Figure 4) to vary with respect to the baseline design by

±50%. While this particular problem could be formulated as a constrained optimisation we shall consider it here as a multi-objective optimisation where both pressure loss and error from the target RTDF are to be improved thus producing a Pareto front from which a designer can make a trade-off between these competing objectives. There are no additional constraints within this optimisation although it could be extended further to consider emissions, manufacturability etc. In summary the design optimisation problem considered here can be defined as,

$$\min\left(\Delta P(\boldsymbol{x}), E_{\mathrm{RTDF}}(\boldsymbol{x})\right) \quad \ldots \quad (1)$$

where the percentage change in pressure loss relative to the baseline design, $\Delta P(\boldsymbol{x})$, is defined as,

$$\Delta P(\boldsymbol{x}) = [P_{\mathrm{Loss}}(\boldsymbol{x}) - P_{\mathrm{Baseline}}]/P_{\mathrm{Baseline}} \times 100 \quad \ldots \quad (2)$$

where,

$$P_{\mathrm{Loss}}(\boldsymbol{x}) = \left(\frac{P_{T_{\mathrm{OGV}}} - P_{T_{\mathrm{Outlet}}}}{P_{T_{\mathrm{OGV}}}}\right) \quad \ldots \quad (3)$$

with $\boldsymbol{x}$ a vector of design variables controlling the dilution port diameters, $P_{T_{\mathrm{OGV}}}$ and $P_{T_{\mathrm{Outlet}}}$ defining the total pressure at the OGV and combustor outlet respectively and P_{Baseline} defining the pressure loss according to Equation 3 for the baseline design. The RMSE to the target RTDF is defined as,

$$E_{RTDF}(\boldsymbol{x}) = \sqrt{\frac{1}{n}\sum_{i=1}^{n}\left(T(\boldsymbol{x}, r_i) - T_{\mathrm{Target}}(r_i)\right)^2} \quad \ldots \quad (4)$$

where, $T(\boldsymbol{x}, r_i)$ defines the RTDF for a given design, $\boldsymbol{x}$, at a set of n fixed radial positions, r_i and where $T_{\mathrm{Target}}(r_i)$ defines the target RTDF at the same set of radial locations.

3.2 Results

A total of nine Latin Hypercube sampling plan points were defined within the design space. These points plus the baseline design define the sampling plan from which the surrogate models are constructed. For each point in the sampling plan the Prometheus input XML file is modified to alter the CAD expressions controlling the diameters of the dilution ports and the path to the resulting fluid volume part file. All other expressions within the input XML file remain unchanged throughout. As per the "geometry centric" philosophy outlined above and illustrated in Figure 2, the CAD environment is therefore generating all of the simulation, pre- and post-processing scripts.

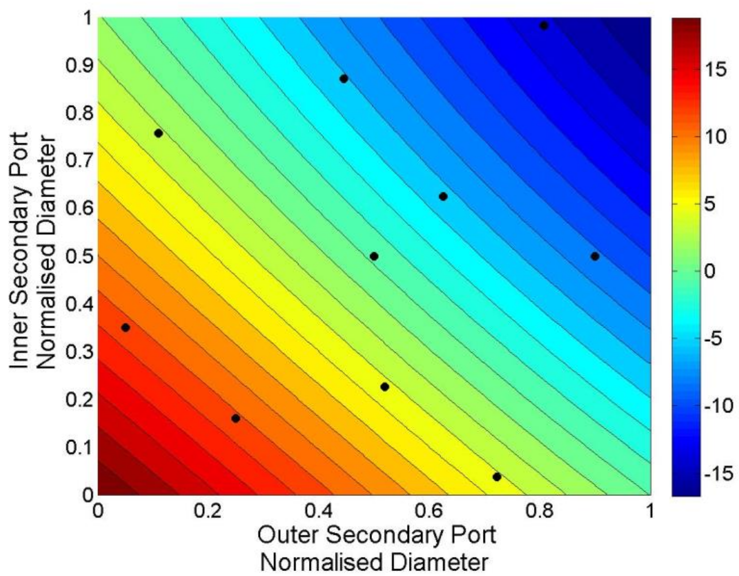

Figure 9 Contours of percentage change in pressure loss predicted from a 10 point sampling plan and an ordinary Kriging based surrogate model

Each combustor design is first analysed using an aerothermal network model, a mesh is then constructed, the CFD simulation run and post-processed as outlined above. The results of these simulations are then used by OPTIMATv2 to construct surrogate models of the percentage change in pressure drop and RMSE to the target RTDF across the two variable design space. Figure 9 illustrates the variation the change in pressure loss while Figure 10 illustrates the variation in the RTDF error predicted by their respective surrogate models.

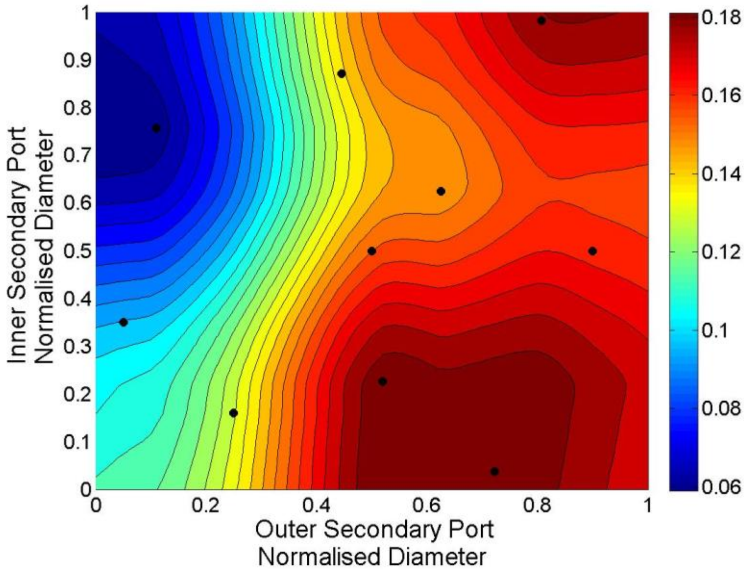

Figure 10 Contours of predicted RMSE to target RTDF profile from a 10 point sampling plan and an ordinary Kriging based surrogate model

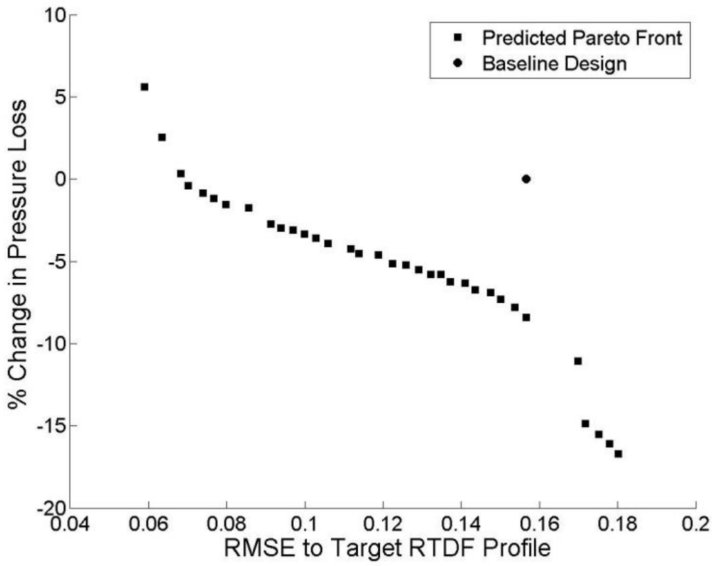

Figure 11 Predicted Pareto front of percentage change in pressure loss versus RMSE to target RTDF profile

The Kriging models presented above are of a relatively high accuracy with a leave-one-out cross-validation predicting r^2 of 0.93 and 0.83 and percentage RMSE of 7.9% and 12.5% for the change in pressure loss and error from target RTDF respectively.

The contours illustrated in Figures 9 and 10 also illustrate that for this set of design variables reducing the pressure loss in the combustor is at odds with matching the target RTDF. Using these surrogate models within a multi-objective optimisation illustrates this further when one examines the resulting Pareto front, see Figure 11. Here the Pareto front predicts an increase of more than 5% over the pressure loss observed by the baseline combustor when conformity to the target RTDF profile is maximised. Similarly it also demonstrates that a substantial reduction in pressure loss is possible at the cost of a considerable loss in RTDF conformity.

As well as illustrating the predicted Pareto front, Figure 11 also illustrates the location of the baseline design with respect to both objectives. This highlights two interesting results, firstly, by constraining the change in pressure loss to be ≤ 0% it is possible to improve conformity in the RTDF by a significant amount from a RMSE of 0.156 to a RMSE of approximately 0.07. Similarly by constraining the RMSE to be ≤ 0.156%, that of the baseline design, we can achieve an 8.3% reduction in pressure loss.

It should be observed, however, that the Pareto front illustrated in Figure 11 is based on the Kriging model predictions and not the results from actual CFD simulations. To validate these results further a subset of the Pareto front should be evaluated using the actual CFD simulations and used to update, or correct, the above surrogate models. This process should continue until convergence or until a predefined number of simulations has been reached. Even without the updating process the above results never-the-less illustrate the ability of the Prometheus system to be effectively employed within a design study or automated optimisation.

4.0 A POTENTIAL MULTI-FIDELITY SURROGATE MODELLING APPROACH

While surrogate modelling approaches can very effectively reduce the number of expensive simulations carried out during an optimisation compared to a direct optimisation algorithm the total simulation cost can still be restrictive. Multi-fidelity surrogate modelling techniques offer a mechanism by which multiple levels of simulation fidelity can be brought together to either enhance a surrogate model's prediction or reduce the number of high fidelity simulations required for an equivalent prediction accuracy. Similar approaches have been used in the past to improve the design optimisations of compressor blades [20] and even combustors [4] (although only using isothermal simulations). Given the expense of the combustor simulations carried out here the idea of augmenting the dataset with additional low fidelity data to reduce the number of CFD simulations is an extremely attractive one.

The Prometheus combustor design system offers a number of different potential sources of lower fidelity data. The XML input file can be easily modified to change the global mesh size and/or the target number of cells across the ports and swirler passages thereby reducing the mesh size and the total simulation time. Following such a strategy does, however, have its potential pitfalls with considerable reductions in fidelity resulting in a growing level of uncorrelation with the high fidelity response. This can lead to a surrogate model which misrepresents the response of the high fidelity design space and can actually mislead the optimisation process [21].

In addition to CFD simulations the Prometheus design system offers another source of performance data that from the aerothermal network model. Let us therefore consider the effectiveness of using the results from these simulations within a multi-fidelity surrogate model prediction of the change in pressure loss. We consider only the change in pressure loss here as the network simulation gives no prediction of the combustor traverse and would therefore offer little insight into the RMSE in the RTDF. As recommended by Toal [20], with such a complete lack of correlation it's better to employ a single fidelity model for this objective. As will be discussed later, this does not necessarily mean, that the absence of a multi-fidelity representation would hamper such an optimisation.

In the following short study we employ a Co-Kriging model [22] to construct a multi-fidelity prediction of the change in pressure loss. In this instance the same 10 point sampling plan used above is employed again but only five of these designs are simulated using high fidelity CFD. This five point subset is selected by defining an optimal spacefilling subset [22]. The Co-Kriging model is therefore constructed from 10 values of pressure loss from aerothermal network simulations and five from CFD simulations.

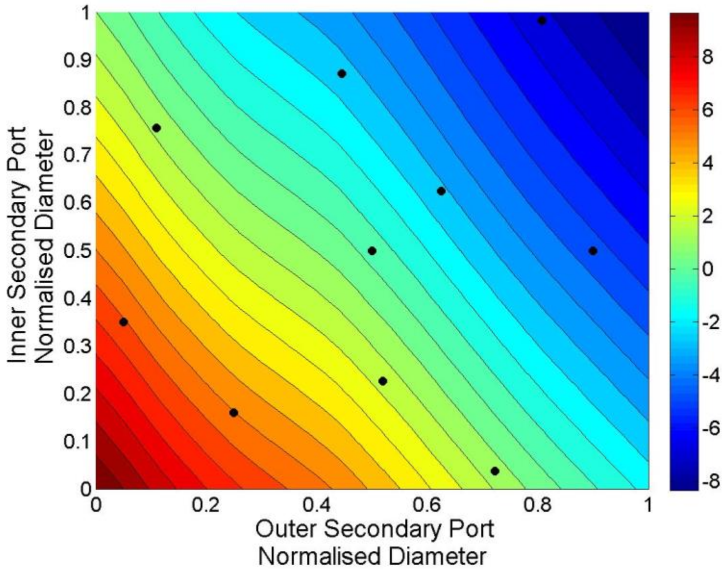

Figure 12 Contours of percentage change in pressure loss predicted from a 10 point sampling plan of aerothermal network results and an ordinary Kriging based surrogate model

The Co-Kriging model is conceptually relatively simple. A Kriging model of the low fidelity data is first constructed. From this differences between the high fidelity results and the low fidelity surrogate model predictions are calculated and a second surrogate model of the differences constructed. The mathematics behind such a model can be found in Forrester et al. [22] for the interested reader. Figure 12 above illustrates the surrogate model constructed from only the aerothermal network results. The model compares very favourably to the equivalent model for the CFD derived data illustrated in Figure 9. The same general trend is present even if the absolute values are not quite the same.

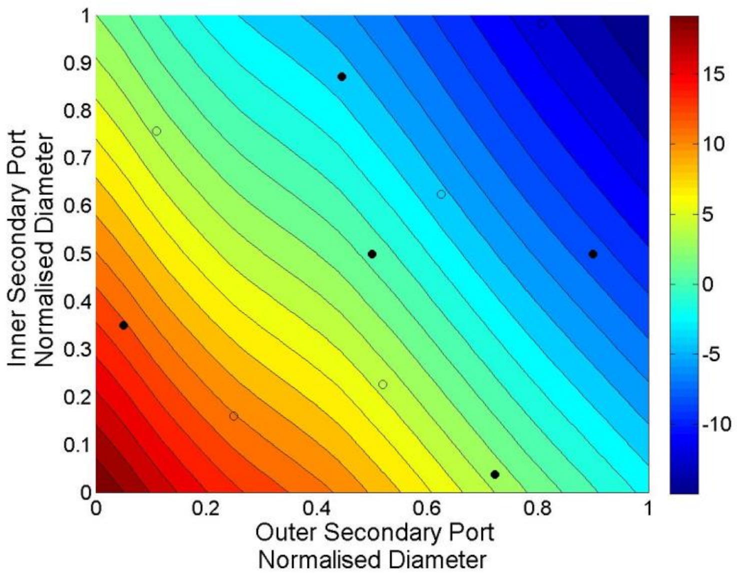

Figure 13 Contours of percentage change in pressure loss predicted from a 10 point aerothermal network and 5 point high fidelity CFD simulation sampling plan combined within a multi-fidelity Co-Kriging based surrogate model

Calculating the error between the surrogate predictions in Figure 12 and the five high fidelity results gives a set of five differences which when a surrogate model is constructed with these points and combined with the low fidelity model in Figure 12 produces the multi-fidelity Co-Kriging prediction of the change in pressure loss illustrated in Figure 13. Here the trend is still well correlated with that of the response in Figure 9, r^2=0.996, but now the absolute ranges are much more aligned, particularly the maximum where there is more high-fidelity data to correct the model.

It should be noted that here we employ the aerothermal network simulation results as our low fidelity data set with the assumption that the high fidelity, CFD derived data, is correct. This would be representative of a preliminary design activity of a novel combustor configuration where the aerothermal network has either not yet been calibrated or cannot be calibrated against experimental data.

In this instance we do not have a multi-fidelity model of the RMSE in RTDF and so potentially the previous optimisation would employ a surrogate model of this objective constructed from only 5 data points. The resulting model is presented in Figure 14 below. While not as accurate as that of Figure 10 the general trend is still present as indicated by a relatively high correlation between the two predictions of r^2=0.911. A multi-fidelity optimisation would therefore be drawn into predominantly the same regions which would then be corrected with subsequent updates to the model. A constrained optimisation would benefit further. As the constraint on pressure loss is accurately modelled the feasible design space is narrowed down very effectively but with a 50% reduction in the number of high fidelity simulations.

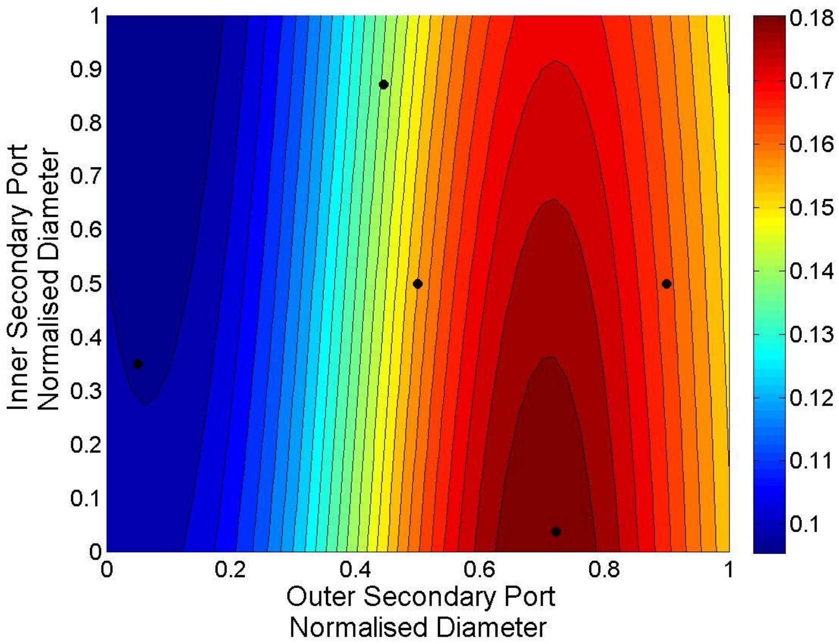

Figure 14 Contours of predicted RMSE to target RTDF profile from a 10 point sampling plan and an ordinary Kriging based surrogate model

5.0 CONLUSIONS

The Prometheus combustor design system offers an alternative method of optimisation workflow construction which has the potential to reduce workflow setup time and rework while giving users the flexibility to explore new concepts. While this approach has been demonstrated with respect to both topological changes and isothermal combustor sub-system optimisations in the past, prior to the current paper it has not been demonstrated within a reacting flow based optimisation.

To demonstrate the system a single skin rich burn combustor is parameterised to permit changes to the diameters of the dilution ports. From this geometry the Prometheus system automatically created an aerothermal network simulation and a CFD simulation, including pre- and post-processing scripts. This capability was used to evaluate 10 different combustor designs spread throughout the design space according to a Latin Hypercube sampling plan. Surrogate models of combustor pressure loss and RMSE to target RTDF were constructed and employed within a multi-objective design optimisation with the resulting Pareto front enabling a designer to trade-off improvements in RTDF conformity against a reduction in pressure loss.

The ability of Prometheus to be employed within a multi-fidelity design optimisation was also explored with a multi-fidelity surrogate modelling strategy mixing aerothermal network and CFD predictions of pressure loss demonstrating a considerable reduction in high fidelity simulation effort with little loss in surrogate accuracy.

While the presented optimisations did not go so far as to include updates to the surrogate models based on the Pareto predictions they never-the-less effectively demonstrate the potential of the Prometheus system when combined with a cutting edge optimisation toolset to improve combustor performance.

There are however, a number of interesting research questions. Can low fidelity CFD simulations be employed within a multi-fidelity framework to aid the RTDF prediction? Could such a process be extended to emissions predictions?

REFERENCES

[1] F. Duchaine, T. Morel and G. M.L., "Computational-fluid-dynamics-based Kriging optimization toole for aeronautical combustion chambers," *AIAA Journal,* vol. 47, no. 3, pp. 631-645, 2009.

[2] J. Rogero and P. Rubini, "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*, Bangalore, 2001.

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

[4] X. Zhang, D. Toal, A. Keane, F. Witham, J. Gregory, M. Ravikanti and M. Zedda, "Isothermal combustor prediffuser & fuel injector feed arm design optimization using the Prometheus design system," *Journal of Engineering for Gas Turbines and Power,* vol. 138, no. 6, pp. 1-19, 2015.

[5] S. Wyse, G. Parks and R. Cant, "Towards the multiobjective optimisation of gas turbine combustors," in *Proceedings of the ASME Turbo Expo*, 2006.

[6] X. Zhang, D. Toal, N. Bressloff, A. Keane and F. Witham, "Prometheus: a geometry-centric optimisation system for combustor design," in *ASME Turbo Expo 2014: Turbine Technical Conference and Exposition*, 2014.

[7] A. Despierre, P. Stuttaford and P. Rubini, "Preliminary gas turbine combustor design using a genetic Algorithm," in *International Gas Turbine and Aeroengines Congress and Exhibition*, Orlando, 1997.

[8] P. Stuttaford and P. Rubini, "Preliminary gas turbine combustor design using a network approach," *Journal of Engineering for Gas Turbines and Power,* vol. 119, no. 3, pp. 546-552, 1997.

[9] S. Stow and A. Dowling, “Thermoacoustic oscillations in an annular combustor,” in *ASME Turbo Expo*, New Orleans, 2001.

[10] S. Stow and A. Dowling, “Low-Order Modelling of Thermoacoustic Limit Cycles,” in *ASME Turbo Expo*, Vienna, 2004.

[11] S. Stow and A. Dowling, “A time-domain network model for nonlinear thermoacoustic oscillations,” *Journal of Engineering for Gas Turbines and Power,* vol. 131, no. 3, 2009.

[12] M. Anand, R. Eggels, M. Staufer, M. Zedda and J. Zhu, “An Advanced Unstructured-grid Finite-volume Design System for Gas Turbine Combustion Analysis,” in *ASME Gas Turbine India Conference*, Bangalore, 2013.

[13] A. Forrester, A. Sobester and A. Keane, Engineering design via surrogate modelling: a practical guide, Chichester: Wiley, 2008.

[14] T. Simpson, J. Peplinski, P. Kock and J. Allen, “Metamodels for computer-based engineering design: Survey and Recommedations,” *Engineering with Computers,* vol. 17, no. 2, pp. 129-150, 2001.

[15] N. Queipo, R. Haftka, W. Shyy, T. Goel, R. Vaidyanathan and P. Tucker, “Surrogate-based analysis and optimization,” *Progress in Aerospace Sciences,* vol. 41, no. 1, pp. 1-28, 2005.

[16] D. Jones, “A Taxonomy of Global Optimization Methods Based on Response Surfaces,” *Journal of Global Optimization,* vol. 21, no. 4, pp. 345-383, 2001.

[17] D. Toal, A. Forrester, N. Bressloff, A. Keane and C. Holden, “An adjoint for likelihood maximization,” *Proceedings of the Royal Society A,* vol. 465, no. 2111, pp. 3267-3287, 2009.

[18] D. Toal, N. Bressloff, A. Keane and C. Holden, “The Development of a hybridised particle swarm algorithm for Kriging Hyperparameter Tuning,” *Engineering Optimization,* vol. 43, no. 6, pp. 675-699, 2011.

[19] K. Deb, Multi-Objective Optimization Using Evolutionary Algorithms, Hoboken: Wiley, 2001.

[20] C. Brooks, A. Forrester, A. Keane and S. Shahpar, “Multi-fidelity Design Optimisation of a Transonic Compressor Rotor,” in *9th European Turbomachinery Conference* , Istanbul, 2011.

[21] D. Toal, “Some considerations regarding the use of multi-fidelity Kriging in the construction of surrogate models,” *Structural and Multi-disciplinary Optimization,* vol. 51, no. 6, pp. 1223-1245, 2015.

[22] A. Forrester, A. Sobester and A. Keane, “Multi-fidelity Optimization Via Surrogate Modelling,” *Proceedings of the Royal Society A,* vol. 463, no. 2088, pp. 3251-3269, 2007.