Abstract – Aerothermal design of turbomachinery components is quite challenging, involving a large number of design parameters, often conflicting multiple objective functions and a number of near equality constraints. Using high fidelity CFD also requires large computing resources. Under stringent time scales, advanced optimisation techniques are needed to achieve a practical design. This lecture will discuss the above issues and the strategies used to achieve a useful application of automatic optimisation to real-world problems. The novel methodology has been included and demonstrated as part of the SOPHY system (SOFT-PADRAM-HYDRA) used by Rolls-Royce, which will be described in this lecture.

2- Introduction

The gas-turbine of a modern jet engine has undergone over 60 years of research and development, hence it can be viewed as a mature technology. Yet the European Union [1] has set very stringent performance improvements to be achieved by 2020; 50% reduction in CO₂ emissions from aircraft per passenger kilometre, halving the perceived aircraft noise, 80% reduction in emissions of NOx, and a five fold reduction in the average accident rate of global operators. To achieve the above, a step change in aircraft and aero-engine technology and science is necessary. Novel design space representation, high-fidelity simulation tools in conjunction with advanced design-space search techniques are needed to improve the performance of a highly complex propulsive system and to minimise the adverse effect of aviation on the environment.

An overview of automatic aerodynamic design optimisation of turbomachinery components was presented in a previous VKI Lecture series by the author [2, 3], and its use in design of a Virtual Engine in an unclassified NATO lecture [4]. They covered some of the most popular methods in parametric design definition, rapid automatic meshing, advanced optimisation methodologies, and high-fidelity CFD based objective definitions. Here a general overview of a practical optimisation methodology is given whereby a large-scale industrial blading problem is solved in an acceptable time frame on a modest cluster.
2.1 Turbomachinery Aerodynamic Design – A Complex Task

As is illustrated in figure 1, the aerothermal design optimisation of a modern gas turbine comprises of many components, such as intake, turbomachinery (fan, compressor, turbines, combustor, seals), exhaust nozzle and nacelle. Arguably, one of the most challenging aspects of design relates to the turbomachinery blading. The aerodynamic design of blading relies on a range of tools from preliminary design analysis, to axisymmetric through-flow methods to the full unsteady 3D Navier-Stokes based CFD solvers. As shown in figure 1, the predictive fidelity of the blade performance increases as the complexity of the flow modelling increases. However, this increase in complexity is usually associated with higher computational costs & run time. It should be emphasised that the current design process makes use of all the listed tools at different, appropriate stages of design. It is not realistic to expect one code to cover all level of geometrical and physical fidelity, however key systems have been developed that efficiently link various simulations codes over a range of applications leading to the Virtual Engine simulation & design capability [4].

Currently the blading design (generating aerofoil profiles) starts in 2D using the data from the through-flow program along the stream sections. The through-flow data is then divided into a number of sections along the span. The design intent is to match the air angles and gas properties designed by the through-flow program. Although, the blade-to-blade design is 2D in essence the variation of the stream-tube height is taken into account for each stream section. The blade-to-blade programs are predominantly based on Euler equations plus an integral boundary-layer method [5] because of transitional boundary layers or 2D Navier-Stokes calculations with some turbulence model closure. These methods are relatively fast on modern computers requiring no more than a few minutes of computing time to simulate the flow through the blade’s passage. Inverse design routines are also available whereby the designer may specify the required surface Mach number distribution and the program calculates the aerofoil shape that delivers the specified flow. However, the accuracy of two-dimensional flow predictions is compromised where there are significant 3D flow features such as secondary flow effects especially near endwalls or strong shock-boundary-layer interactions, as shown in Fig. 2. These flow features are three-dimensional in nature and can only be accurately predicted by solving the three-dimensional Navier-Stokes equations.

Figure 1: Evolution of design-optimisation versus the computing power needed – possible number of design iterations.

Figure 2: Transonic Flow through a 2D Turbine Stage (MT1) - Gradient of density contours.