Proceedings of ASME Turbo Expo 2015: Turbine Technical Conference and Exposition
GT2015
June 15 – 19, 2015, Montréal, Canada

GT2015-42249

MULTI-OBJECTIVE CFD OPTIMISATION OF SHAPED HOLE FILM COOLING WITH MESH MORPHING

D. Proietti
University of Rome
"Tor Vergata"
Rome, Italy

A. Pranzitelli
ANSYS UK Ltd.
Sheffield, UK

G.E. Andrews
Energy Research Institute
University of Leeds,
Leeds, UK

M.E. Biancolini
University of Rome
"Tor Vergata"
Rome, Italy

D.B. Ingham
ETII, University of Leeds
Leeds, UK

M. Pourkashanian
Energy Research Institute
and ETII, University of Leeds,
Leeds, UK.

ABSTRACT

A Computational Fluid Dynamics (CFD) optimisation of a single row of film cooling holes was performed. The aim was to achieve the highest adiabatic cooling effectiveness while minimising the coolant mass flow rate. The geometry investigated by Gritsch et al. [1] was the baseline model. It consisted of a row of cylindrical, 30^o inclined holes, with a mainstream inlet Mach number of 0.6, a blowing ratio of 1 and a plenum for the upstream cooling air flow. The predictions agreed with the experimental data with a maximum deviation of 6%. The geometry was then optimised by varying three shape parameters: the injection angle, the lateral hole expansion angle and the downstream compound hole angle. A goal driven optimisation approach was based on a design of experiments table. The minimisation of the coolant mass flow together with the maximisation of the minimum and average cooling effectiveness were the optimisation objectives. The shape modifications were performed directly in the ANSYS Fluent CFD solver by using the software RBF Morph in the commercial software platform ANSYS Workbench. There was no need to generate a new geometry and a new computational mesh for each configuration investigated. The dependency of the average effectiveness along the plane centreline on the three geometrical parameters was investigated based on the metamodel generated from the design of experiments results. The goal driven optimisation led to the optimal combination of the three shape parameters to minimise the coolant flow without reducing the cooling effectiveness. The best results were obtained for a geometry with 20^o hole angle and 7.5^o compound angle injection, leading to a reduction of 15% in the coolant mass flow rate for an enhanced adiabatic cooling effectiveness. The results also showed the preponderance of the centreline angle over the other two parameters.

NOMENCLATURE

D	Film cooling hole diameter, m
L	Film cooling hole length, m
M	Blowing ratio = $\rho_c u_c / \rho_f u_f$
M_m	Mach number of the mainstream jet
T_c	Coolant temperature, K
T_s	Adiabatic Surface temperature, K
T_r	Mainstream recovery temperature, K
T_m	Mainstream temperature, K
u_c	Coolant mean velocity in the hole, m/s
u_f	Crossflow flame or hot gas velocity, m/s
X	Downstream distance from trailing edge of hole
X/D	Dimensionless downstream distance from the trailing edge of the hole
α	Angle of the film cooling hole centreline to the outlet wall
ρ_c	Coolant density, kg/m^3
ρ_f	Flame or hot gas crossflow density, kg/m^3
η	Adiabatic Cooling effectiveness = $(T_s - T_r)/(T_c - T_r)$

INTRODUCTION

Turbine blade and combustor wall cooling is crucial to the operation of modern gas turbines due to the use of gas temperatures that are much higher than the melting point of the metal walls. Future high performance turbines will operate at higher turbine inlet temperatures. This will require higher film cooling effectiveness, together with improvements in the wall material operating temperatures.

Blade cooling has three main components: back side convective cooling, internal passage convective cooling, and film cooling. Coolant air for the film cooling is bled from the compressor and flows through the blade metal passages to form a film of cold air between the metal and the hot gas cross flow. For combustors, film cooling does not have any adverse effect on the cycle thermal efficiency, but for turbine blades the compressor air bleed has not been heated by combustion and, therefore, there is a loss in efficiency as the expansion across the turbine blade is reduced and compression work is not fully recovered. This loss in cycle efficiency increases as the turbine film cooling mass flow increases. However, the gain in efficiency, from the ability to operate at higher turbine inlet temperatures, leads to an overall cycle efficiency gain. Unfortunately, this blade cooling air makes the combustor operate richer with higher NO_x, and in the limit, the combustor will approach stoichiometric conditions before the overall engine equivalence ratio is stoichiometric (which is the highest thermal efficiency condition).

Eventually, the total mass flow of coolant used for blade film cooling will become the limiting factor in reaching the highest cycle thermal efficiency. Therefore, reducing this film coolant mass flow is beneficial for maximum cycle efficiency. In combustor wall cooling, air mass flow used for film cooling is required to be minimised. This is because any air used for film cooling is not available in the low NOx primary zone of the combustor and as a consequence this operates hotter with a penalty in NOx formation.

There are two ways of reducing the coolant mass flow rate: improving the adiabatic effectiveness of film cooling [2] through the hole outlet shape design and improving the internal wall cooling. The internal wall is effectively a heat exchanger with a hot metal that is cooled by the air passing through the array of holes. Increasing the hole angle for a fixed wall thickness increases the hole length to diameter ratio, L/D, which potentially increases the internal wall cooling. The increase in hole area with change in its expansion angle might reduce the internal hole heat transfer through a reduction in the Re. The present work only considered the optimisation of the adiabatic film cooling effectiveness and it is possible that this is not the optimum for the overall cooling effectiveness. If additional internal wall cooling was added, such as impingement cooling, this would not influence the optimum design of the film cooling, as the impingement jet is always used to cool the midpoint between the film holes.

Previous studies [1, 3-7] have shown that the jet exit hole shape plays an important role in determining the adiabatic cooling effectiveness. This term is often used as a quantitative measure of the film cooling performance in insulating the hot crossflow from the cooled wall. In addition, the exit hole shape will also have an influence on the internal wall heat transfer, as well as on the coolant spread in the axial and lateral directions. Due to a film cooling jet 'lift-off' effect a pair of counter-rotating vortices occur in the near field of a hole exit, the 'cooled' jet flow tends to be 'separated or detached' from the blade surface and this causes a significant reduction of the film cooling effectiveness.

Other factors such as blowing ratio and the operating conditions of hole pressure loss and coolant temperature also influence the cooling effectiveness. The traditional film cooling configuration uses a cylindrical tube that links an internal cooling duct to the external cross flow domain. The jet hole shape becomes elliptical at the exit and its size and the longitudinal axis length and angle influence the internal metal wall cooling as well as the external film cooling. The injection angle is normally around 30^{o}, as for the reference case used in this work [1]. Goldstein et al. [2] were among the first to pioneer the use of shaped film holes for improved film cooling performance. The performance of inclined holes with a 10 deg laterally flared exit was compared with the performance of streamwise inclined cylindrical film holes. Adiabatic cooling effectiveness data showed that the shaped film hole provides better lateral coverage and better centre line effectiveness. This was part of the reason for using only the centreline adiabatic cooling effectiveness in the optimisation process.

Makki and Jakubowski [3] presented downstream heat transfer results for a film hole with a trapezoidal shaped expansion. They showed that the shaped film hole consistently provided better heat transfer characteristics than simple cylindrical holes with the same metering section. They found that the shaped holes offered up to 23% better film cooling performance than the corresponding cylindrical hole. Schmidt et al. [4] and Sen et al. [5] presented two companion papers in which the effect of adding a 15 deg forward diffusion exit to a streamwise oriented hole was investigated. They found that the film hole exit diffusor demonstrated better spread of adiabatic effectiveness than the cylindrical counterpart. From the heat transfer coefficient standpoint, the forward expanded hole performed poorly, presumably because of the increased interaction between the jet and the mainstream. Hyams et al. [6] studied the effects of slot jet shaping on the heat transfer downstream of a slot jet. They found that shaping of the slot inlet and exit provided significant gains in the film cooling performance.

In experiments, both infrared image and transient liquid crystal schemes have been commonly employed to obtain the surface temperature and adiabatic cooling effectiveness. When computational simulation is conducted, selections of turbulent models may affect the results. The effects of turbulence modeling on film cooling simulations have been investigated by a number of researchers. For example, the V2F k–ε turbulence model was employed by Jia et al. [7], the standard k–x model by Brittingham and Leylek [8], and k– ε model by Heidmann et al. [9]. Recently, Tyagi and Acharya [10] employed a large eddy simulation (LES) scheme to investigate the detailed coherent flow structures of film cooling. Numerical simulation can provide ideal boundary conditions but may fail to accurately predict the flow separation and correct physics. Nevertheless, most published work did not employ the real gas turbine operating conditions at high temperature and magnitude.

Simulation driven design is growing as an effective approach toward complex industrial and research applications. A very well established calculation workflow consists in the set-up of a high fidelity Computational Fluid Dynamics (CFD) model that has first to be validated (against experiment if available and/or with convergence checks). A baseline model can then be used to predict the effect of hole shape using mesh morphing to improve the baseline design. For the latter task metamodelling is nowadays a standard practice. This involves the evaluation of system response using a Design of Experiment (DOE) Table interpolation to determine the optimal configuration.

Parameterization of the film hole shape is an ideal application of mesh morphing CFD models. New shapes are generated by deforming the mesh of the baseline CFD model, i.e. just updating nodal positions, which requires a negligible computational time compared to any remeshing procedure. Importantly, preserving the same mesh structure eliminates the remeshing noise that can be confused with the effect of the design parameters.

Several algorithms have been explored for this task. A common and well-established technique, the Free Form Deformation (FFD) [11] method, deforms volumes and controls their shape using a trivariate Bernstein polynomial. The method is meshless, so it can be easily implemented in parallel partitioned meshes with hybrid elements. It allows the definition of new interesting shapes but it lacks accurate local surface control. Such accurate control can be achieved using mesh-based methods, for example in the pseudosolid method [12], where an elastic FEM solution is used to propagate the deformation inside. Parallel implementations can in this case be difficult and extra effort is required when surface movements are not known in advance.

The meeting point between these two approaches can be achieved using Radial Basis Functions (RBF) interpolation, which combines the benefits of a meshless method with great precision. In this case the RBF morphing field is interpolated using a cloud of points with given displacements. Even if there is interesting research demonstrating that RBF can be successfully adopted for the deformation of CFD meshes [13, 14], their numerical cost has limited their application in the past (direct solution grows by N^3 where N is the number of RBF centres).

The first industrial implementation of RBF mesh morphing was introduced in 2009 with the software RBF Morph [15] that comes with a fast RBF solver for the bi-harmonic kernel which performances scales as $N^{1.6}$. A system of radial functions is used to produce a solution for mesh movement/morphing from a list of source points and their displacements. This approach is valid for both surface shape changes and volume mesh smoothing. Radial bases are able to interpolate everywhere in the space a function defined at discrete points giving the exact value at original points. The behaviour of the function between points depends on the kind of basis adopted. The radial function can be fully or compactly supported; in any case, a polynomial corrector is added to guarantee compatibility for rigid modes.

The use of RBF requires the solution of a linear system of order equal to the number of source points introduced for coefficient calculations. Once the unknown coefficients are calculated, the motion of an arbitrary point inside or outside the domain (interpolation/extrapolation) is expressed as the summation of the radial contribution of each source point (if the point falls inside the influence domain). A complete description of the tool is given in [16] while examples of applications can be found in Caridi and Wade [17], Cella and Biancolini [18], Khondge and Sovani [19] and Biancolini et. al [20].

In this paper CFD is used to investigate adiabatic film cooling hole outlet design improvements that will reduce the mass flow of the film cooling air without reducing the film cooling effectiveness. The adiabatic wall experiments of Gritsch et al. [1] were used as a reference case. After the validation of the numerical model, a design of experiments approach was used for the optimisation. Three geometrical parameters were investigated: the injection angle, the lateral expansion angle and the downstream compound angle. The geometrical modifications were obtained by morphing the computational mesh by using the RBF Morph software integrated in the commercial CFD solver ANSYS Fluent as an add-on. The optimisation procedure was developed within the ANSYS Workbench environment [21].

REFERENCE CASE

For the purposes of this study, the hot rig results of Gritsch et al. [1] were modelled and considered as a reference case for the optimisation. The experimental apparatus is schematically shown in figure 1. Table 1 shows the operating conditions of the experiments that were reproduced in the CFD model.

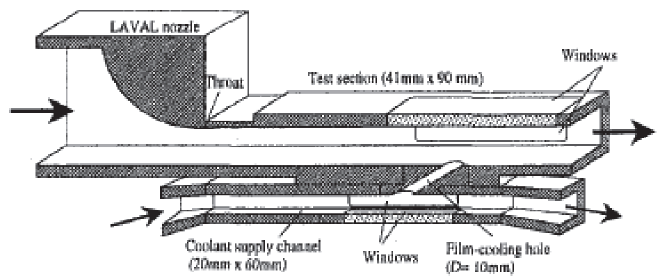

Fig. 1: Experimental apparatus [1].

Table 4 Operating conditions of the reference case

Coolant temperature	T_c	290 K
Blowing ratio	M	1
Temperature ratio	T_c/T_m	0.54
Coolant Mach number	M_c	0.0
Mainstream Mach number	M_m	0.6

NUMERICAL MODEL AND VALIDATION

The numerical model reproduces the main features of the experimental apparatus [1]. Due to the symmetry of the problem, only half of the geometry was modelled. The boundary zones are shown in Fig. 2. A pressure inlet and a pressure outlet were imposed for the hot stream channel, with a fixed pressure ratio. The test Mach number was obtained by means of an adequate pressure difference. The test condition of pressure plenum in the cold stream channel was reproduced assuming the same pressure for inlet and outlet. A solid wall, modelled as adiabatic for the validation, separates the two channels. A symmetry boundary condition was set at the two sides of the model. No-slip wall boundary conditions were imposed for all the solid walls of the geometry.

To ensure the grid independency of the solution, three different meshes were made, i.e. coarse, medium and fine, made of 300K, 1M and 1.4M cells, respectively. The medium

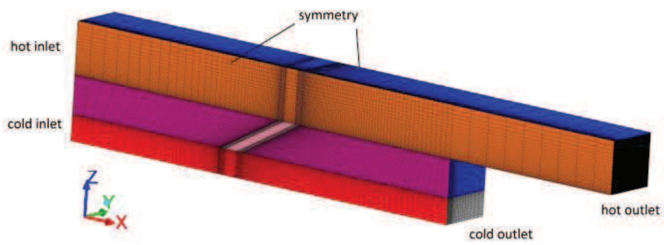

Fig. 2: Computational domain

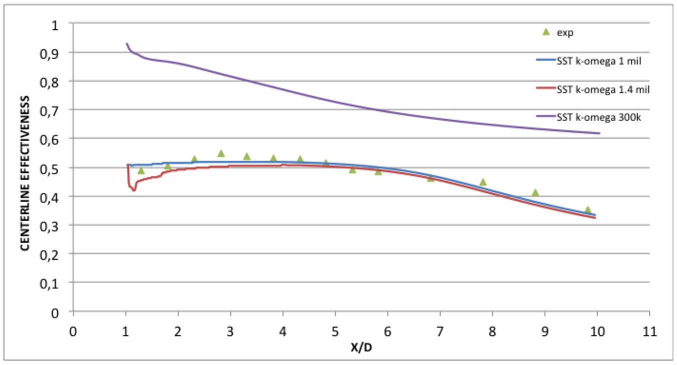

Fig. 3 Comparison between the different meshes results of centreline adiabatic film cooling effectiveness

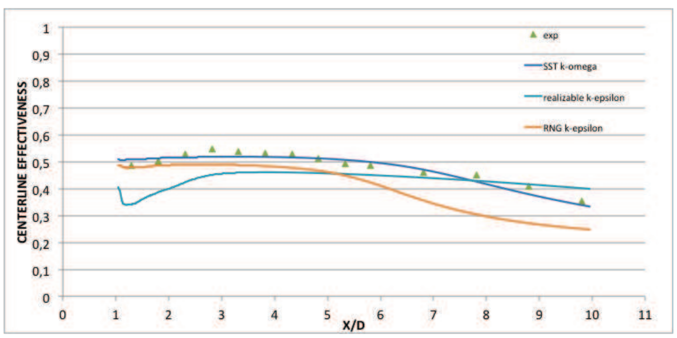

Fig. 4 Comparison between the different turbulence model results of centreline adiabatic film cooling effectiveness

and fine meshes were obtained by successively refining the coarse mesh in the hole region and in the near wall region of the plate downstream of the hole. The grid independency study was performed using the SST k-ω turbulence model. The parameter considered for the validation was the centreline adiabatic cooling effectiveness, measured in the downstream hole region.

Oguntade [21, 22] examined the influence of hole outlet geometry using fixed geometry mesh CFD with adaptive grids and compared the centreline and radially averaged adiabatic cooling effectiveness computations for the measurements of Sinha et al [23] as a reference case. A range of turbulence models were used and the ranking of best agreement for centreline and surface averaged was the same. Oguntade et al. [21, 22] also showed that the predictions tended to overpredict the centreline in the far distance from the hole, but underpredicted the area between rows of holes and several other CFD modellers of this geometry had also found this. The radial variation of cooling effectiveness was not well predicted in the far distance, but was better close to the hole. Ogundtade et al. [21, 22] showed that the predicted benefit of improved hole outlet shape was the same trend for the centreline and radially average predictions and the best geometry found experimentally could be predicted using either centreline or radially averaged adiabatic cooling effectiveness. Thus, it was considered that for design optimisation, the centreline adiabatic cooling effectiveness could be used and this was computationally simpler.

The results are shown in Fig.3 which shows that the coarse mesh predictions overestimated the measured adiabatic cooling effectiveness. The results obtained with both the medium and fine meshes were in good agreement with the experimental data. The fine mesh predictions had a maximum deviation of 6% from the experimental data. As the results obtained with the medium mesh does not differ significantly from the fine mesh, the medium mesh was used for the optimisation process.

To choose the optimal turbulence model for the case considered, the realisable k-ε, the RNG k-ε and the SST k-ω turbulence models were compared. The results are shown in Fig. 4 where the SST k-ω model was shown to have a very good agreement in the whole computational domain. Both the k-ε models showed an underestimation of the cooling effectiveness in the near and far regions of the domain.

SHAPE MODIFICATIONS AND OPTIMISATION PROCEDURE

In order to maximise the cooling effectiveness and minimise the coolant flow rate, three shape modifications were considered: the variation of the injection angle, the lateral expansion angle and the compound angle of the hole. The modification of the injection angle consisted of the rotation of the axis of the complete hole, in both negative and positive directions, keeping plane the inlet and outlet surfaces to evaluate the effect of the hole angle in the film coolant jet spreading. Figs. 5 and 6 show the effect of the hole axis rotation.

Modifying the lateral expansion angle consisted in the creation of a compound angle in the perpendicular direction with respect to the flow in order to increase the width of the hole outlet and obtain a better lateral coverage of the surface. The modification was applied only in the final region of the hole, to allow the flow to develop inside the hole. Figs. 7 and 8 show the modification of the lateral expansion angle.

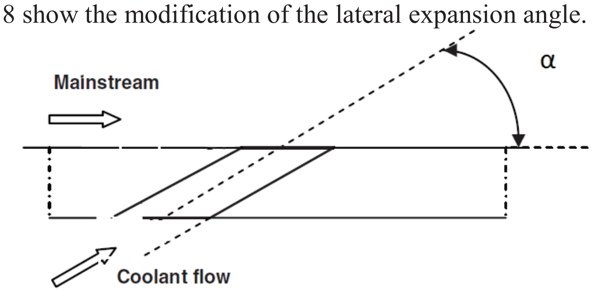

Fig. 5 Injection angle scheme

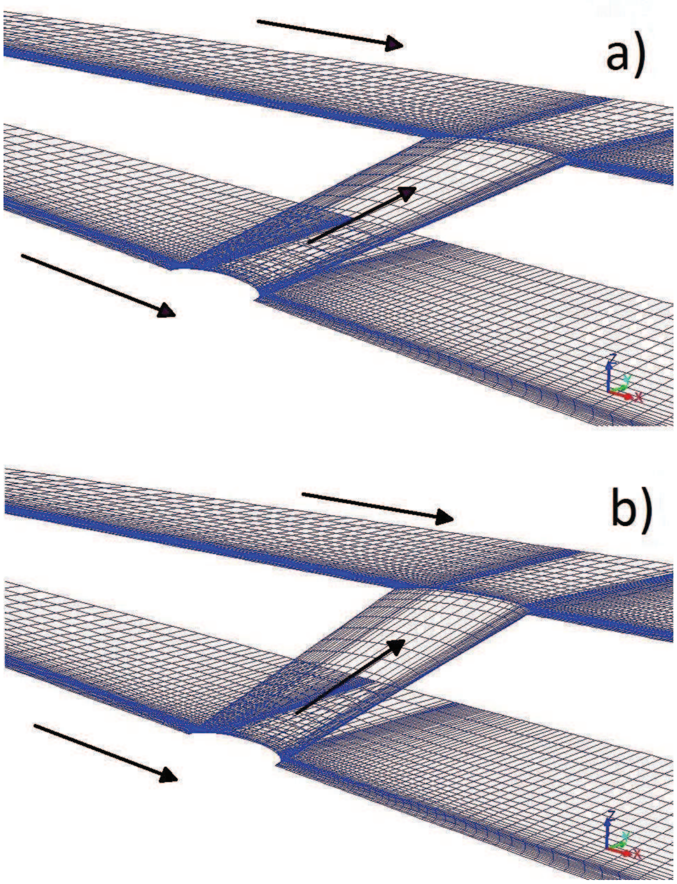

Fig. 6 Hole axis rotation from 20 degree (a) to 35 degree (b). The black arrows indicate the direction of the flow.

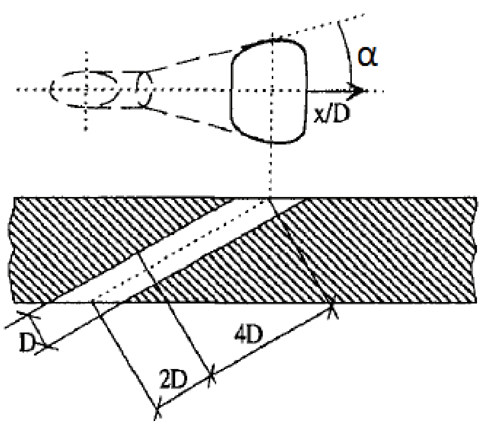

Fig. 7 Lateral expansion scheme

Modifying the downstream compound angle leads to an increase in the length of the hole outlet. Having a smoother angle at the exit was predicted to reduce the separation of the flow from the surface. The modification was applied only at the final region of the hole, to allow for the development of the flow inside the hole which would maximise the internal hole heat transfer. Figs. 9 and 10 show such a modification.

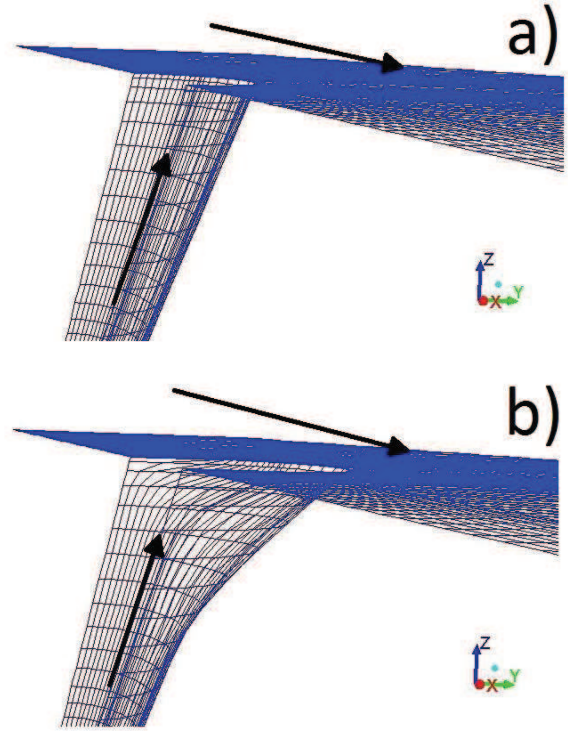

Fig. 8 Lateral expansion from the 0 degree of the baseline (a) to 10 degree (b). The black arrows indicate the direction of the flow.

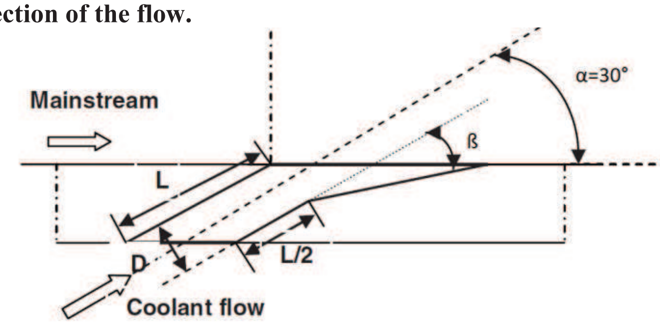

Fig. 9 Downstream compound angle scheme

The geometrical modifications were obtained by morphing the computational grid within the CFD solver environment of ANSYS Fluent using the RBF-Morph solver, without the need to regenerate the geometry and remeshing. The angle variations were obtained by displacing a number of control points while constraining others. Amplification factors were given to a base displacement in order to obtain the desired angular variation.

A design of experiments was used to automatically generate a set of simulations which cover the design space specified by the parameters' range shown in Table 2. Fifteen design points were generated, as shown in Table 2, using the Optimal Space-Filling Design (OSF) [24] method. The data from the design of experiments simulations was then used to generate a response surface metamodel from which the performance of the other designs and the effects of the geometrical modifications on the output parameters can be predicted. The response surface is also used to predict the

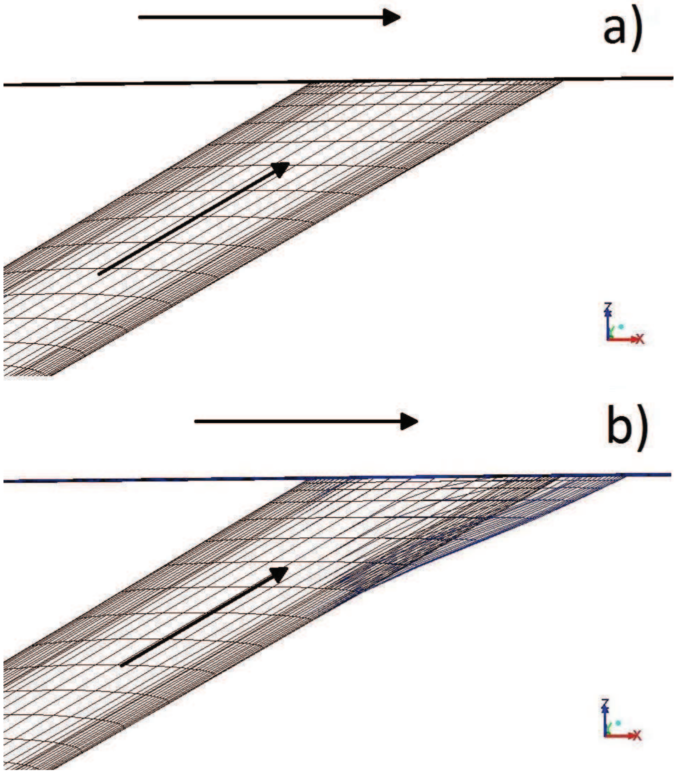

Fig. 10 Downstream compound angle variation from 0 degree of the baseline (a) to 10 degree (b). The black arrows indicate the direction of the flow.

Table 5 Ranges of variation of the optimisation parameters from the baseline

	From (deg)	To (deg)
Injection angle	20	35
Lateral expansion angle	0	10
Compound angle	0	10

combination of geometrical modifications that give the optimum design.

The monitored parameters were:

• average centreline effectiveness along the centreline of the plate from the hole up to 100 mm downstream of the hole;

• minimum centreline effectiveness along the centreline of the plate from the hole up to 100 mm downstream to the hole;

• coolant mass flow rate through the hole.

The optimisation targets were the maximisation of the average and minimum effectiveness and the minimisation of the coolant flow rate.

The convergence was checked by monitoring the velocity at two locations downstream of the hole and the average value of the centreline effectiveness. The baseline geometry solution was used as the initial condition for all the design of experiments calculations. All the design points were run for a number of iterations to achieve convergence. To estimate the number of iterations needed, a calculation of a design point

Table 6 List of the design points considered

DP	Injection angle (deg)	Lateral angle (deg)	Compound angle (deg)
1	34.5	0.68	4.26
2	33.5	5	6.71
3	32.5	3.5	1.12
4	31.5	9.5	3.5
5	30.5	1.12	8.53
6	29.5	7.6	0.31
7	28.5	0.06	1.64
8	27.5	5.85	9.5
9	26.5	2.85	2.85
10	25.5	1.64	0.06
11	24.5	8.53	5.03
12	23.5	0.3	5.85
13	22.5	6.7	0.7
14	21.5	4.26	7.6
15	20.5	2.22	2.22

was monitored and convergence was obtained after about 500 iterations. The design points were then conservatively run for 1000 iterations. The simulations were performed on a quad-core, 3.2GHz CPU.

RESULTS

From the analysis of the response surface it is possible to investigate the effects of the geometrical parameters on the adiabatic film cooling efficiency and the coolant mass flow rate. Fig 11 shows the high sensitivity of the coolant flow rate, for constant adiabatic average centreline cooling effectiveness, to the rotation. A significant reduction of the coolant flow rate as the injection angle increased was found. The compound angle contributes to the reduction of the flow rate only for low injection angles, which is when the hole tends to become vertical. This is because with inclined holes and film coolant flow attached to the wall, there is no advantage of the downstream compound angle. This is because a downstream compound angle is used to help to keep an inclined jet coolant film attached to the wall and is ineffective if the jet is attached to the wall. For a detached jet with a lower inclined hole angle Fig. 11 shows that there is then an advantage of the downstream compound angle.

Fig. 12 shows the effects of the two expansion angles on the adiabatic cooling effectiveness for constant coolant mass flow. It can be seen that they improve the effectiveness if increased singularly, while the effect becomes less significant when increased together. Fig. 12 shows that the peak adiabatic cooling effectiveness occurs for no downstream compound angle and the maximum lateral angle. The later angle increases the outlet area of the hole and changes it alignment to that of a slot jet. The effect is for the film to attach to the surface and give a high adiabatic cooling effectiveness. There is then no advantage of having a downstream compound angle.

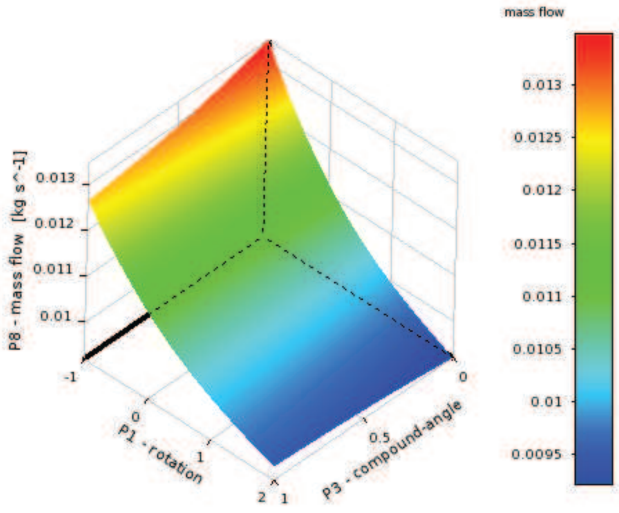

Fig. 11 Response surface for the coolant mass flow versus injection angle (rotation) and compound angle. Angles given in terms of amplification factors.

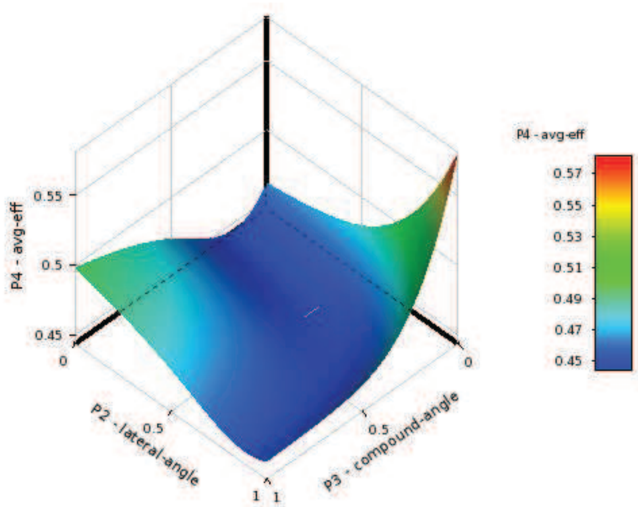

Fig. 12 Response surface for the average adiabatic effectiveness versus lateral-angle and compound angle. Angles given in terms of amplification factors.

For lower lateral jet spread angle the film cooling jet is lifted off the surface and the application of a downstream compound angle has some benefit in reducing the flow separation.

A number of candidate design points as optimum were obtained from the response surface. After verification by CFD calculations, two candidates were selected. The combinations of the input parameters are shown in Table 4, along with the resultant output parameters and the percentage variation with respect to the baseline geometry.

Both the design points give the same increase of average effectiveness (10%). The design point A produced a higher increase of minimum effectiveness with a slight increase of the coolant flow rate, while the design point B produced a lower increase of minimum effectiveness with a significant reduction of coolant mass flow rate. Therefore, the design point B is advisable when the main objective is the reduction of the coolant flow rate. Fig. 13 shows the distribution of adiabatic cooling effectiveness downstream of the cooling hole for the candidate B against the baseline design.

Figs. 14 and 15 compare the temperature distribution and the cooling air concentration of candidate B against the baseline, respectively. It can be seen that the optimal configuration proposed produces a higher concentration of coolant near the wall ensuring a lower temperature as a consequence. Figs. 14 and 15 clearly show that the baseline film lifts off the surface at the hole exit and has poor cooling performance in the hear hole region. The optimised jet with reduced air mass flow and lower angle has attached coolant flow in the near hole region.

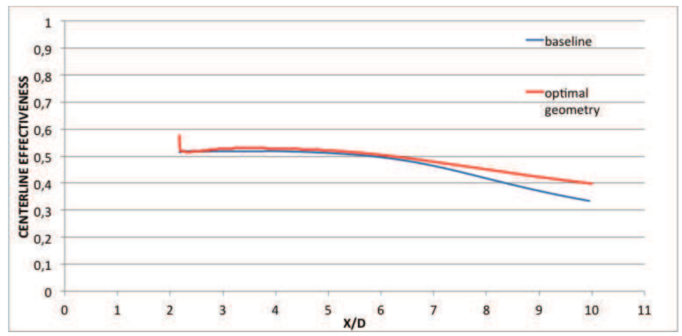

Fig. 13 Comparison of centreline adiabatic effectiveness between baseline (blue line) and optimal geometry (red line)

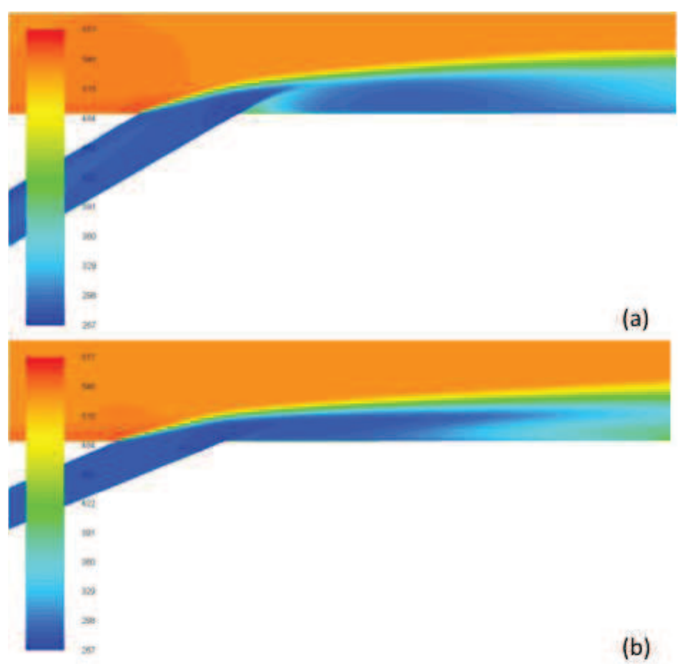

Fig. 14 Comparison of temperature distribution between the baseline (a) and optimal geometry (b)

Table 7 Optimal design points. The variations are relative to the baseline case

Design point	Axis rotation (deg)	Lateral angle (deg)	Compound angle (deg)	Average effectiveness	Minimum effectiveness	Coolant mass flow (kg/s)
A	20	10	0	0.53 (+10%)	0.45 (+43%)	0.0098 (+1.25%)
B	22	0.5	7.5	0.52 (+10%)	0.39 (+26%)	0.0083 (-14.56%)
Baseline	30	0	0	0.48	0.32	0.0097

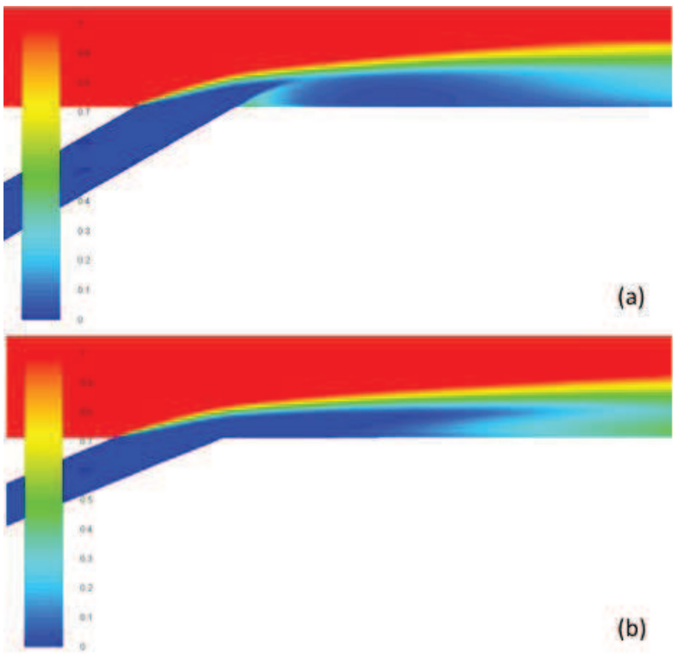

Fig. 15 Comparison of cooling air concentration between baseline (a) and optimal geometry (b)

CONCLUSIONS

A CFD design approach to single row film cooling geometry optimisation was carried out using mesh morphing. Improved adiabatic cooling effectiveness using a lower air mass flow proportion were the optimisation targets for which the hole geometry shape was changed. Previous work in the literature investigated different shaped hole outlet configurations by creating different geometries and remeshing, and considering a fixed geometry.

The problem was investigated by a shape parametric CFD model in the Fluent environment, which is able to consider the simultaneous effect of the shape parameters that can be varied continuously and seamlessly over a wide range.

Starting from a cylindrical 30 degree inclined hole configuration, the geometry was optimised by varying three shape parameters, i.e. the injection angle, the lateral expansion angle and the downstream compound angle. The adiabatic wall experimental results of Gritsch et al. [1] were used for validation of CFD results. The shape modifications were performed as mesh modifications directly in the CFD solver by the RBF Morph software. The numerical procedure was automated within the ANSYS Workbench environment. With a relatively small number of directly calculated design points a response surface metamodel was created, allowing a sensitivity study of the output parameters on the geometrical modifications.

A goal driven optimisation approach based on a design of experiments was used, assuming the minimisation of coolant flow and the maximisation of average and minimum coolant effectiveness as targets.

The results showed that there were two important design factors: hole injection aangle and coolant air flow reduction. The best optimised geometry was predicted to have 15% reduction in the coolant mass flow rate with an injection angle reduced from 30^o to 22^o. The optimised design also had a lateral expansion angle of 0.5^o and hole compound angle of 7.5^o.

The CFD optimisation procedures used in the present work are suitable for a range of optimisation prediction in gas turbine film cooling. The angle of the jet in the transverse direction and the separation of jets in the axial direction for multi-row film cooling could both be optimised using the present techniques. The optimisation of trench outlet designs are all areas where the present procedures could predict optimised designs, which could replace the present rather intuitive experimental approach to hole outlet film cooling design. Also the optimisation techniques could be extended to the overall cooling effectivess by adding conjugate heat transfer to compute the internal wall cooling.

ACKNOWLEDGEMENTS

We would like to thank the University of Leeds for provision of computational facilities and ANSYS UK for the provision of time for AP. D. Proietti would like to thank the European Union for funding for a research exchange.

REFERENCES

[1] M.Gritsch, A.Schulz, S.Wittig, 1998, “Adiabatic Wall Effectiveness Measurements of Film-Cooling Holes With Expanded Exits” ASME J.Turbomach., 120, pp. 149- 156.

[2] R.J. Goldstein, “film cooling”, 1971, Advances in Heat Transfer 7.

[3] Y.Makki, G.Jakubowski, 1986, “An experimental study of film cooling from diffused trapezoidal shaped holes”, AIAA paper no. 86-1326.

[4] D.Schmidt, B.Sen, D.Bogard, 1996, “Film cooling with compound angle holes:adiabatic effectiveness”, ASME J.Turbomach., 118(1196) pp. 807- 813.

[5] D.Schmidt, B.Sen, D.Bogard, 1996, “Film cooling with compound angle holes: heat transfer”, ASME J.Turbomach., 118(1996) pp. 800-806.
[6] D.Hyams, J.H.Leylek, 1996, “Effects of geometry on slot-jet film cooling performance”, ASME paper no. 96-GT-187.
[7] R. Jia, B. Sunden, P. Miron, B. Leger, 2003, “Numerical and experimental study of the slot film cooling jet with various angles”, Proceedings of the ASME Summer Heat Transfer Conference, pp. 845–856.
[8] R.A. Brittingham, J.H. Leylek, 2002, “A detailed analysis of film cooling physics: Part IV – Compound-angle injection with shaped holes”, ASME J. Turbomach. 122 pp. 133–145.
[9] J.D. Heidmann, D. Rigby, A.A. Ameri, 2000, “A three-dimensional coupled internal/ external simulation of a film-cooled turbine vane”, J. Turbomach. 122, pp. 348–359.
[10] M. Tyagi, S. Acharya, 2003, “Large eddy simulation of film cooling flow from an inclined cylindrical jet”, ASME J. Turbomach. 125 (4) pp.734–742
[11] Sederberg TW, Parry SR., 1986, “Free-form deformation of solid geometric models.” The proceedings of the 13th annual conference on computer graphics and interactive techniques (SIGGRAPH ’86). ACM, New York, NY, USA, pp. 151–60.
[12] Masud A, Bhanabhagvanwala M, Khurram RA., 2007, “An adaptive mesh rezoning scheme for moving boundary flows and fluid-structure interaction”, Comput Fluid 36(1), p. 77–91.
[13] de Boer A, van der Schoot MS, Bijl H., 2007, “Mesh deformation based on radial basis function interpolation” Comput Struct 85(11–14) p. 784–95.
[14] Jakobsson S, Amoignon O., 2007, “Mesh deformation using radial basis functions for gradient based aerodynamic shape optimisation”, Comput Fluid 36(6), p. 1119–36.
[15] Biancolini ME, Biancolini C, Costa E, Gattamelata D, Valentini PP., 2009, “Industrial application of the meshless morpher RBF morph to a motorbike windshield optimisation.” The proceedings of the European automotive simulation conference (EASC). Munich. Germany.
[16] Biancolini ME., 2012, “Mesh morphing and smoothing by means of radial basis functions (RBF): a practical example using Fluent and RBF morph. Handbook of research on computational science and engineering: theory and practice”, IGI Global.
[17] Caridi D, Wade A., 2012, “Higher-speed CFD. Professional motorsport magazine”, www.pmw-magazine.com, p. 56.
[18] Cella U, Biancolini M., 2012, “Aeroelastic analysis of aircraft wind-tunnel model coupling structural and fluid dynamic codes”, J Aircraft 49(2), p.407–14.
[19] Khondge A, Sovani S., 2012 “An accurate, extensive, and rapid method for aerodynamics optimisation: the 50:50:50 method” SAE Technical Paper, 2012-01-0174.
[20] M.E. Biancolini et al. 2014, “Sails trim optimisation using CFD and RBF mesh morphing”, Computers & Fluids 93 pp. 46–60.
[21] Oguntade, H.I., Andrews, G.E., Burns, A., Ingham, D., and Pourkashanian, M., 2010. “CFD Predictions of Single Row Film Cooling with Inclined Holes: Influence of Hole Outlet Geometery”. Proc. ASME Turbo Expo, Glasgow. ASME Paper GT2010-22308.
[22] Oguntade, H.I., Andrews, G.E., Burns, A., Ingham, D. and Pourkashanian, M., 2011, “Improved Trench Film Cooling with Shaped Trench Outlets”. Proc. ASME Turbo Expo, Vancouver. ASME Paper GT2011-45253. Journal of Turbomachinery 135 (2), 021009, 2012. Paper TURBO-11-1201. Doi:10-1115/1.4006606.
[23] Sinha, A.K., Bogard, D.G. and Crawford, M.E., 1991. “Film-Cooling Effectiveness Downstream of a Single Row of Holes with Variable Density Ratio”. Journal Turbomachinery, 113, 442-229.
[24] ANSYS Workbench manual, ANSYS, Inc. 2013.

