Mesh Sensitivity of RANS Simulations on Film Cooling Flow

Yuewen Jiang* Alexander Murray Luca di Mare Peter Ireland

Oxford Thermofluids Institute, University of Oxford, Osney Mead, Oxford OX2 0ES, England

**email: yuewen.jiang@eng.ox.ac.uk*

Abstract

Accurate prediction of film cooling flow is a necessity in the design of high-pressure turbine components given the temperature of hot gas exceeds the melting point of the component material to achieve high thermal efficiency. Nowadays, Computational Fluid Dynamics (CFD) methods are becoming increasingly popular to predict film cooling flows. When compared to large eddy simulation (LES) and direct numerical simulation (DNS), Reynolds Averaged Navier-Stokes (RANS) method is most frequently employed due to its much lower computational cost, though it struggles to accurately solve the highly three-dimensional and anisotropic cooling flows. However, few publications have focused on the mesh sensitivity problem of RANS simulations. And yet mesh generation is fundamental to CFD simulations and directly affects the accuracy of solution. Two typical film cooling hole geometries, namely cylindrical and fan-shaped, are utilised to investigate the mesh sensitivity of RANS simulations with low, medium, and high blowing ratios. Seven mesh sizes ranging from super coarse to super fine (4000 times denser) are employed for analysis. In general, the comparisons show that the computational results are close to the experimental film feature and averaged effectiveness while mesh convergence becomes challenging for RANS modelling, especially for the fan-shaped hole.

Keywords: mesh sensitivity, mesh convergence, film cooling, heat transfer, Reynolds Averaged Navier-Stokes (RANS), Computational Fluid Dynamics (CFD)

Highlights:

- RANS solutions with a medium size mesh are close to the experimental data;
- The surface film effectiveness of the cylindrical hole converges well, though the 3D spatial variables are challenging;
- Mesh convergence of fan-shaped hole is more difficult than that of a cylindrical counterpart.
- Mesh convergence at high blowing ratios provides a greater challenge than at lower blowing ratios;

1. Introduction

Turbine entry temperature (TET) is continuously increasing in pursuit of improving the thermal efficiency of the gas turbine and jet engine. Currently, temperatures already exceed the melting temperature of the blade materials [1]. Cooling technologies are necessary in permitting turbine components to survive the high-temperature hot gas and provide adequate component lifespan. Film cooling provides one such major cooling method. Cooling is achieved through the formation of a thin, low-temperature film between the hot mainstream flow and turbine component thereby protecting the surface. An accurate prediction of blade temperature is critical for the design process. Blade lifespan may be reduced by up to 50% by under-predicting the temperature by 30 K [2]. However, the film cooling flow is challenging to accurately predict due to the inherent physical complexities surrounding boundary layer modelling, heat transfer, turbulence mixing, and separation amongst others.

The physics surrounding jet flow is an established research topic. Fric and Roshko [3] studied the structure of jet flow formed by the interaction of a turbulent jet into a uniform stream. The flow was fundamentally different from the phenomenon of vortex shedding from a solid bluff body. The wake structure was orderly and the corresponding wake Strouhal number of 0.13 was most sharply defined for jet/crossflow velocity ratios near the value 4.

The mixing characteristics associated with a round jet injected into cross-flow was quantified in [4]. The comparisons demonstrated that jet-to-cross-flow density ratios below unity tended to mix less well than for uniform density conditions, demonstrated to result from differences in the nature of higher-density cross-flow entrainment into lower-density shear layer vortices. The near-field flow developments of jets in cross-flow was investigated by Zang and New [5]. Both laser-induced fluorescence and particle-image velocimetry measurements were employed. It was observed that there existed substantial flow shear stresses as the pair of inner vortices interacted. A comprehensive review of the individual flow features of steady jets interacting with a cross-flow was provided by Mahesh [6].

Film cooling flows are typically studied via either experiments or CFD. The Pressure Sensitive Paint (PSP) technique was employed to study the effect of rotation on leading-edge film cooling of a turbine blade [7]. It was found that the rotation speeds significantly changed the film cooling traces. High-resolution infrared measurements were used to study the inlet geometry of a laidback fan-shaped film cooling hole [8]. The comparisons showed that the inlet geometry of the cooling hole significantly affected cooling performance. One of the advantages of experimental research is the physical flow provided by the wind tunnel, although the veracity of the results is dependent upon the accuracy of the measurement technique. An alternative to conducting experiments is the use of CFD simulation which have become increasingly popular with the fast development of computer science and CFD methods. The use of CFD allows for the prediction of a detailed flow-field aiding understanding of the physics behind the cooling flow. However, film cooling flows still provide a significant challenge for CFD to efficiently and accurately predict.

Large-eddy simulations (LES) were used to study round jet flow in [9]. The obtained mean and turbulent statistics provided good agreement with experimental data. Large-scale coherent structures were accurately reproduced in the simulations. Direct numerical simulations (DNS) were employed in [10] to study a round turbulent jet. Comparisons showed good agreement between computational results and experimental data. Examination of the turbulent kinetic budgets showed that the near field was far from a state of turbulent equilibrium. The results were used to comment upon the difficulty involved in predicting this flow using RANS computations. The flow structures of an inclined round jet were investigated in [11]. Results showed that the blowing ratio affected the cooling jet flow structure significantly. The effect of vortical structures on the film cooling effectiveness was also studied. DNS was employed to study the effects of mainstream Reynolds number on film cooling flows employing a fan-shaped hole [12]. Higher mainstream Reynolds numbers were observed to improved the film cooling effectiveness with a corresponding decrease in hairpin vortices. Mainstream turbulence intensity was also observed to affect the film cooling effectiveness. A round film cooling hole embedded in a contoured crater was investigated by Kalghatgi and Acharya [13] who comprehensively studied the unsteady flow physics of film cooling. Time-averaged and instantaneous flow structures were shown in an effort to investigate the mechanisms of anti-counter rotating vortex pairs and attenuating counter-rotating vortex pairs. The modal analysis showed a loss of coherence and increased time scales of shear layer structures as the crater depth was increased.

Given the constraints on computational resources and simulation times, LES and DNS are still unfeasible for simulating the film cooling flow with many holes, even considering their benefits in providing more accurate results and greater physical understanding of the flow fields. Realistic configurations of high-pressure turbines require hundreds of cooling holes per blade. The complex geometries and high Reynolds numbers are considerably challenging for LES and DNS methods. Therefore, RANS methods remain the major CFD technique utilised for film cooling studies even though there is still an associated time cost when compared to some model-based methods [14, 15].

Garg [16] utilised a 3D Navier-Stokes code to simulate the heat transfer coefficient on a film-cooled rotating turbine blade by using Wilcox's $k-\omega$ turbulence model, Coakley's $q-\omega$ model, and the zero-equation Baldwin-Lomax (B-L) model. The results indicated that the $k - \omega$ model compared most favourably with the experimental data. Hoda and Acharya [17] investigated seven different turbulence models for the computation of film cooling flow. They observed that close agreement with experimental results was obtained at the jet exit and far downstream of the jet injection region, but all models typically over-predicted the magnitude of the velocities in the wake region behind the jet. Azzi and Lakehal [18] compared the performance of two classes of turbulence models for reproducing near-wall cooling flow physics and heat transfer, namely, DNS based two-layer approaches and quadratic and cubic explicit algebraic stress formulations. The comparison demonstrated that only the anisotropic eddy-viscosity/diffusivity model could correctly predict the spanwise spreading of the temperature field and reduced the strength of the secondary vortices.

Miao and Ching [19] used CFD methods employing the standard $k-\varepsilon$ turbulence model to predict the 3D flow and heat transfer of a concave plate which was cooled by two rows of film-cooling holes. The effect of Reynolds number, angular locations of the two rows of film holes and blowing ratio were investigated. It was found that the laterally averaged film cooling effectiveness over the concave surface with a straight-blow plenum was slightly greater than that of a cross-blow plenum at all test blowing ratios. Han et al. [20] utilised the Reynolds stress model to predict the film cooling effectiveness and the associated heat transfer coefficient in a 1-1/2 turbine stage. The authors' observed that the tilted stagnation line on the rotor leading edge moved from the pressure side to the suction side, and the instantaneous coolant streamlines shifted from the suction side to the pressure side with increasing rotational speed. The Reynolds-averaged Navier-Stokes equations closed by the $k-\omega$ SST turbulence model was used to investigate a film cooling geometry with high hole pitch to diameter ratio [21]. The experimental data was successfully reproduced using the CFD method.

Ma et al. [22] studied the effect of cooling flow on a transonic squealer tip using both experiments and CFD. Two turbulence models, namely the SpalartAllmaras (SA) model and $k-\omega$ SST model, were employed. The trends obtained from both experiment and CFD compared favourably. The $k-\omega$ SST model demonstrated a greater ability in capturing the flow physics when compared to the SA model. Yan et al. [23] investigated the heat transfer and film cooling characteristics on a squealer-winglet blade tip by utilising the $k-\omega$ turbulence model. The comparison indicated that the $k-\omega$ model showed better overall agreement with experiment than the other models investigated. Four turbulence models, namely the Realizable $k-\varepsilon$ model, Standard $k-\omega$ model, Shear Stress Transport (SST) model, and transitional SST model, were compared in [24]. It was observed that the Realizable $k-\varepsilon$ model showed the strongest agreement with experimental data.

Different cooling configurations, ranging from a single hole to two rows of multiple holes, were systematically studied using RANS in [25]. The $k-\omega$ SST turbulence model was employed in the simulations. A strong, non-linear effect, resulting from the interaction between coolant jets was demonstrated. The study indicated that surface curvature and rotation significantly affected the film cooling effectiveness and jet interactions. Ravelli and Barigozzi [26] utilised the heat/mass transfer analogy in an investigation on leading-edge film cooling on a first-stage vane. The Realizable $k-\varepsilon$ turbulence model with enhanced wall treatment was used as the turbulence closure. Murray et al. [27] applied CFD simulations to understand the flow-field in double-wall effusion, cooling geometries. The computational area-weighted averaged overall film cooling effectiveness agreed well with experimental data. Chen et al. [28] used the CFD method with the $k-\omega$ SST turbulence model to investigate the cooling flow. The pressure obtained by the CFD simulations and experimental data were in good agreement.

Despite the remarkable success of RANS applications on film cooling simulations, few publications concentrate on one of the fundamental problems of CFD methods — mesh sensitivity. Mesh convergence is a fundamentally important consideration in CFD simulations. The accuracy of CFD solutions would be expected to improve when the mesh resolution is increased. The predicted result gradually converge to a certain value as the mesh spacing is decreased.

However, mesh convergence remains one of the great challenges when RANS method is used to solve the complex turbulence beyond the capability of its mathematical model. For example, a relatively small junction separated-flow region can cause a serious mesh convergence issue that was concentrated by AIAA Drag Prediction Workshop series already held six times from 2001 to 2016 [29–31]. Moreover, film cooling flow is a well-known jet-in-crossflow problem. It is highly three-dimensional and anisotropic such that RANS modelling is typically inadequate in accurately solving the flow [32].

Due to the reality RANS method is widely used on film cooling flow simulations, it is therefore necessary to particularly study the mesh sensitivity. A single film hole is explored such that an extremely fine mesh much denser than that used by LES can be generated for the detailed comparisons. It is systematically investigated for different blowing ratios via a typical flat plate film cooling configuration with both cylindrical and fan-shaped holes.

2. RANS Method

The integral form of Reynolds Averaged Navier-Stokes (RANS) equations can be written as follows:

$$\frac{\partial}{\partial t}\iiint_{\Omega}\mathbf{Q}dV+\iint_{\partial\Omega}\mathbf{F}(\mathbf{Q})\cdot\mathbf{n}dS=\iint_{\partial\Omega}\mathbf{F}^{vis}(\mathbf{Q})\cdot\mathbf{n}dS \tag{1}$$

where t stands for time. Ω and $\partial\Omega$ are the space and boundary of a discrete control volume respectively. $\mathbf{n}$ denotes the unit outward-normal vector of the control volume face $\partial\Omega$. V is the volume of Ω. S represents the area of

$\partial\Omega$. $\mathbf{Q}$ is conservative flow variable vector. $\mathbf{F}(\mathbf{Q})$ and $\mathbf{F}^{vis}(\mathbf{Q})$ are inviscid flux vector and viscous flux vector, respectively. Meanwhile, the turbulent viscosity is solved by the widely-used $k-\omega$ SST turbulence model [33].

The finite volume method is used for the discretization of the Navier-Stokes (NS) Equations (1). A 2D example of control volumes m and n is shown in Figure 1. Suppose that control volume n is the p-th neighbor of control volume m. The semi-discrete form of the NS equations for control volume m can be written as

$$\frac{\mathrm{d}(\mathbf{Q}_m V_m)}{\mathrm{d}t} + \sum_{p\in\Xi(m)} \mathbf{F}(\mathbf{Q}_{m,p})\cdot\Delta\mathbf{S}_{m,p} = \sum_{p\in\Xi(m)} \mathbf{F}^{vis}(\mathbf{Q}_{m,p})\cdot\Delta\mathbf{S}_{m,p} \tag{2}$$

where $\Delta\mathbf{S}_{m,p}$ is the area vector of the p-th face of control volume m.

The Roe's scheme [34] is used to calculate the inviscid fluxes

$$\mathbf{F}(\mathbf{Q}_{m,p}) = \mathbf{F}^{Roe}(\mathbf{Q}^L_{m,p}, \mathbf{Q}^R_{m,p}) \tag{3}$$

where $\mathbf{Q}^L_{m,p}$ and $\mathbf{Q}^R_{m,p}$ are the left- and right-hand side values of the face, respectively. For the second-order finite volume method, both are linearly reconstructed through the gradient of the control volume. For a flow variable q, the reconstruction is given by

$$\begin{cases} q^L_{m,p} = q_m + \phi_m[\nabla q_m \cdot (\mathbf{x}_{m,p} - \mathbf{x}_m)] \\ q^R_{m,p} = q_n + \phi_n[\nabla q_n \cdot (\mathbf{x}_{m,p} - \mathbf{x}_n)] \end{cases} \tag{4}$$

where ∇q represents the flow gradient obtained by the Green-Gauss method. $\mathbf{x}_{m,p}$ is the coordinate vector of the face center. $\mathbf{x}_m$ denotes the coordinate vector of the centroid of control volume m. Subscript n stands for the p-th neighbor of control volume m. ϕ is the limiter function.

The viscous fluxes are discretized by using the central difference scheme. The numerical methods are implemented in the Rolls-Royce in-house code HYDRA which is a node-based finite volume solver and has been validated for both aerodynamics and heat transfer applications [25, 35–39]. All the presented simulations are fully turbulent and steady, and wall functions are not used.

3. Geometry and CFD Setup

The test cases chosen here are based on four principles:

- The geometry is representative of typical film cooling configurations;
- The geometry is simple such that it is easy to generate a high quality mesh;
- Structured mesh is preferred because it allows us to change the mesh resolution in non factor of two refinements while creating meshes of similar local quality;
- Flow conditions, such as Reynolds number, density ratio, and blowing ratio, are representative of typical values employed in the film cooling of high-pressure turbine components.

A flat plate with a single hole is therefore employed to study the mesh sensitivity of RANS simulations on film cooling simulations. The geometry and flow conditions are based on the experiment reported in [40]. Two typical hole geometries are used in the present study, namely cylindrical and fan-shaped holes depicted in Figures 2 (a) and (b). The hole diameter D is 10 mm. Hole length is 60 mm. Thus, hole length-to-diameter ratio is 6. And the inclination angle of the hole is 30°. For the fan-shaped hole, the diffusion angle is 14°.

The computational domain is shown in Figures 2 (c) and (d). Mainstream flow Mach number is 0.3 and the Reynolds number is 32000 based on the film hole diameter of 10 mm. Inlet turbulence intensity is 5.2% while the boundary layer thickness is 0.065D. Inlet boundary conditions for the mainstream channels are specified as total pressure and temperature, namely, 1 bar and 540 K, respectively. The total temperature at inlet of the coolant channel is 340 K while the total pressure varies to match the specific blowing ratios. Inlet turbulence intensity of the coolant is also 5.2%. Moreover, all walls are specified as adiabatic conditions.

Meshing is the foundation of the numerical discretization of the complex NS equations. The accuracy of CFD simulations is significantly affected by mesh size, mesh quality, numerical methods and turbulence among others. Only the effect of mesh size with similar quality is studied in this paper. It is therefore critical to define the rules of the mesh refinement for the mesh sensitivity study. Ideally, the mesh cells can all be locally scaled by the same factor while maintaining the mesh quality. To this end, the study employs a hexahedral mesh which is generated via a multi-block structured mesh. The basic partition of blocks depicted in Figure 3 is maintained whilst the mesh is refined or coarsened. Changes in the structured mesh size can maintain good local spacing scaling in each direction (x, y, and z). Thus a family of self-similar meshes are generated. A fine mesh is shown in Figure 4 for both cylindrical and fan-shaped cooling holes. In the boundary layer, there are 46 nodes with a y^+ of 0.2 and height ratio of 1.1 at the wall normal direction.

4. Results and Discussion

Details of the flow-field, heat transfer, and film cooling effectiveness are analyzed for the different mesh sizes. Three typical blowing ratios (low, medium, and high) for each hole geometry are investigated. The blowing ratio (M) is defined by

$$\mathrm{M} = \frac{\rho_c u_c}{\rho_m u_m} \tag{5}$$

where ρ is the density and u denotes the velocity. The subscript c and m represent the coolant flow and mainstream, respectively. It should be noted that the density and velocity of coolant are measured as the cross-section averaged value inside hole. For the fan-shaped hole with non-constant cross-section, these quantities are evaluated in the cylindrical part.

Seven mesh sizes ranging from super coarse to super fine are employed for the mesh sensitivity study. The number of mesh nodes in the finest mesh is more than 4000 times of that of the coarsest one. For the cylindrical hole, the coarsest mesh uses only 6 nodes around the cooling hole while the finest mesh utilises 472 nodes/hole. The intermediate mesh sizes are 14 nodes, 28 nodes, 58 nodes, 118 nodes, and 236 nodes around the film hole. The corresponding sizes of entire mesh nodes are 0.012 million (M) (super coarse), 0.042M (extra coarse), 0.161M (coarse), 0.670M (medium), 2.94M (fine), 12.4M (extra fine), and 52.4M (super fine), respectively. It should be noted that the mesh size suitable for a LES simulation is around 10 million for this kind of cooling flow.

For the fan-shaped hole, only 10 nodes are utilised around the film hole in the coarsest mesh with 656 nodes employed for the finest mesh. Between these extremes, the other meshes use 20 nodes, 40 nodes, 80 nodes, 164 nodes, and 328 nodes around the film hole. The respective mesh sizes are 0.016M (super coarse), 0.057M (extra coarse), 0.242M (coarse), 0.980M (medium), 4.10M (fine), 17.3M (extra fine), and 72.8M (super fine), respectively.

The meshing is based on the fine mesh by coarsening & refining the spacing at walling normal, streamwise, and lateral directions. The factor of mesh nodes change is 20% at wall normal direction while it is 2 at the others. The interval mesh size is changed by the factor of around 4 instead of 4.8 because the O-type topology is used around the film hole. It should be noted that the y^+ of the coarsest mesh is 1 and that of the finest one is 0.07.

4.1. Basic Flow Features of Film Cooling

The CFD results obtained using the fine mesh shown in Figure 4 is utilised for the below discussion. The basic features of film cooling flow are demonstrated in Figures 5 and 6. The figures demonstrate a parallel plane 0.002D from the film cooled wall surface, along with seven perpendicular slices spaced equally downstream of the cooling hole. Specifically, these are located 1D, 3D, 5D, 7D, 9D, 11D, and 13D downstream of the hole where the distance is measured from the centre of the leading edge and trailing edge of the cooling hole exit. Thus the position of 1D is located at the trailing edge of hole exit. Meanwhile, there is blanking in the lateral and normal directions.

The non-dimensional vorticity in the wall normal direction is compared for both cylindrical and fan-shaped holes in Figure 5. The non-dimensional ω' of vorticity ω is given by

$$\omega' = \omega \frac{\mathrm{D}}{U} \tag{6}$$

where U represents the mainstream flow speed. The colourbar legend is displayed at the bottom of images. For the cylindrical hole, a counter rotating pair with opposite vorticity can be seen at a blowing ratio (M) of 0.5. The strength of these vortices decreases as the coolant moves downstream. As the blowing ratio is increased, the strength of these vortices is seen to also increase. This trend is also observed in the parallel plane of the hole exit due to the higher momentum coolant jet. Furthermore, the high momentum jet also causes a small accompanying vortex pair near the viscous wall. For the fan-shaped hole, the cooling film becomes much wider with a corresponding reduction in the strength of vorticity when compared with that of the cylindrical hole. Moreover, there now exist two pairs of opposite vorticity. The strength of these is weak when the blowing ratio (M) is 0.5 with a rapid decay as the coolant moves downstream. The higher blowing ratio significantly enhances the strength of the vorticity downstream.

The non-dimensional temperature field is demonstrated in Figure 6. The non-dimensional temperature defect is given by

$$s(T) = \frac{T_r - T}{T_r - T_c} \tag{7}$$

where T_r is the recovery temperature of hot gas; T stands for the local temperature; and T_c denotes the coolant temperature. A pair of vortices is clearly exhibited for the cylindrical hole as shown in Figures 6 (a), (c), and (e). The vortex pair is observed to be approximately symmetrical about the hole centreline. For a blowing ratio (M) of 0.5 demonstrated in Figure 6 (a), the size of the vortices increases significantly as the film travels downstream from positions 1D to 5D. At this blowing ratio, it is found that the coolant jet adheres well to the wall such that it is protected from the hot gas. For blowing ratios (M) of 1.0 and above, due to the higher coolant jet momentum after ejection, the distance over which the size of vortices grow increases when compared with that of the lower blowing ratios. Moreover, the lowest temperature moves away from the wall such that the wall temperature is higher than that of the lower blowing ratio.

For the fan-shaped hole, contours and streamlines at three blowing ratios are shown in Figures 6 (b), (d), (f). The film coverage is visibly improved compared with those obtained from the cylindrical hole, with a much wider low-temperature region observable. The effect of blowing ratio on the vortices and temperature is again demonstrated, however, the features are different from those of cylindrical hole. The increase of blowing ratio not only increases the size of vortices but also changes the locations of vortices which move laterally due to the enhancement of coolant momentum with a more complex flow pattern.

4.2. Mesh Sensitivity of Film Effectiveness

Figures 7 and 8 demonstrate the mesh sensitivity of adiabatic film cooling effectiveness for cylindrical and fan-shaped holes, respectively. The film cooling effectiveness (η) is defined by

$$\eta = \frac{T_{un} - T_w}{T_{un} - T_c} \tag{8}$$

where T_{un} represents the adiabatic wall temperature without cooling, T_w signifies the local wall temperature with cooling and T_c denotes the temperature of coolant. On the uppermost row, the number n/hole represents the number of mesh nodes around a film cooling hole and N M denotes the number of entire volume mesh nodes with M standing for million nodes. The experimental images of film cooling effectiveness are shown in the leftmost column. Both lateral and streamwise directions are normalized by the hole diameter D.

4.2.1. Film Effectiveness of the Cylindrical Hole

Figure 7 shows the mesh sensitivity of film cooling effectiveness for the cylindrical hole. For the case of the coarsest mesh, there is only one cell on the wall surface downstream of the cooling hole with 6 nodes around the cooling hole. The mesh is too coarse to provide an accurate assessment of the effect of varying film blowing ratios with little variation in cooling effectiveness observed as the blowing ratio was increased, and little observed decay in film cooling effectiveness downstream. The results are much improved when the number of nodes around the cooling hole is doubled. In this case, the film is observed to differ as the blowing ratio is varied, even though the results are still inaccurate. The effect of blowing ratio becomes clearer still when the mesh is further refined. The film cooling effectiveness at varying blowing ratios demonstrates different features.

For a blowing ratio (M) of 0.5, the result obtained by the coarsest mesh fails to show the decay of film cooling effectiveness. Meanwhile, the coolant lift-off phenomenon is observed for the solution obtained by the meshes with 14 nodes and 28 nodes around the cooling hole. The coolant is seen to lift off at the point of ejection before reattaching downstream. This incorrect trait is not observed in the result solved by the super coarse mesh. It indicates that a finer mesh may obtain a worse film trend when compared to a coarser mesh. That demonstrates that refining a mesh does not always result in a more accurate cooling film solution. The mesh convergence of film images performs favourably. The film solved by the mesh with 58 nodes/hole is in close agreement to that captured by the finer meshes. Generally, the centreline effectiveness of CFD solutions is observed to be greater than that of experiments, with films generally demonstrating less spanwise diffusion.

At a blowing ratio (M) of 1.0, the phenomenon that the coolant lifts off at the point of ejection before reattaching downstream is successfully captured by all the mesh sizes. However, the point of reattachment exhibits great discrepancy. When the mesh is refined, this point gradually moves downstream then moves upstream. The solution by 28 nodes/hole appears excessively far from the cooling hole. The trend from the simulated film approaches that of experimental measurements when the mesh size is increased to 12.4 million. It indicates that capturing jet lift-off presents a significant challenge for RANS simulations, with high mesh resolution required. Furthermore, the film detail is still changed when the mesh is further refined to 472 nodes/hole.

When the blowing ratio (M) is further increased to 1.5, the trend of mesh size effect is similar to that of blowing ratio 1.0. The super and extra coarse meshes significantly over-predict the film effectiveness. The reattachment point is also captured. But this phenomenon is not clearly observed by the other meshes. On the contrary, these solutions demonstrate a strong lift-off flow feature and a little film is solved. Meanwhile, the comparison indicates that a mesh independent film solution is not achieved for this high blowing ratio, even though the solution obtained by the mesh of 12.4 million nodes is close to that by 52.4 million.

In summary, the comparisons between images of experimental film effectiveness and those obtained by the simulations for the fine mesh, indicate that the basic flow features are in good agreement for the low and medium blowing ratios investigated. The film cooling effectiveness decay rate is captured by the simulations as the coolant moves downstream. For medium blowing ratio, the feature of lift-off at jet ejection and reattachment downstream is captured and the reattached point is obtained accurately. However, it is observed that the width of the simulated film is narrower than that of the experiment for the high blowing ratio. The film effectiveness is also slightly lower.

4.2.2. Film Effectiveness of the Fan-shaped Hole

The mesh sensitivity of film cooling effectiveness obtained by the fan-shaped hole for different blowing ratios is shown in Figure 8. For the coarsest mesh with only 10 nodes around the film hole, the solved film cooling effectiveness does respond to an increase in blowing ratio. The high mass flow rate of coolant appears to enhance the effectiveness. This phenomenon is different from that obtained by the cylindrical hole with the coarsest mesh. It also is able to capture the decrease in film cooling effectiveness at a blowing ratio of 0.5 as flow moves downstream. However, the mesh independent solution of local film cooling effectiveness becomes difficult to achieve. The mesh refinement is observed to change the local film distribution even when the mesh size is already large.

For the blowing ratio (M) of 0.5, the coarsest mesh only utilises 10 nodes around the film hole and therefore is not able to adequately represent the geometric complexity of the fan-shaped hole. The solution is dramatically improved when it is refined to 20 nodes/hole with the hole geometry becoming much more accurate. The obtained flow pattern becomes much closer to that of the experimental observation. However, two wakes of the film can not be observed at both sides when the mesh is refined to 40 nodes around the film hole. As it is further refined to 80 nodes/hole, two wakes of the film are evidently solved and the decay of film cooling effectiveness is much closer to that of the experimental measurement. The local film cooling effectiveness seems convergent when the mesh is refined to 236 nodes/hole.

For the blowing ratio (M) of 1.5, the decay of film cooling effectiveness is captured by the coarsest mesh with only 10 nodes around the film hole. This trend is adequately enhanced when the mesh is further refined to 20 nodes per hole. But this coarse mesh still can not well obtain two wakes of the film until the mesh is refined to 40 nodes/hole. The film obtained by a mesh with 80 nodes/hole is similar to that by 328 nodes/hole. However, that solved by the further finer mesh with 164 nodes per hole becomes apparently different and the effectiveness is significantly over-predicted. The trend indicates that once again, mesh independence can not be confirmed even if the films solved by the two largest mesh sizes are close.

For the blowing ratio (M) of 2.5, the mass flow rate of coolant becomes high, thus the momentum is well enhanced. When the mesh is refined to 40 nodes around the film hole, it successfully captures this influence of mass flow rate. However, the film cooling effectiveness still appears excessively great with a slow decay rate and it is apparently wider that of the finer meshes. When the mesh is further refined, the wakes of the film becomes narrower compared to those at a blowing ratio of 1.5. Meanwhile, the width of the entire film is also slightly decreased. It is found that the film effectiveness significantly decreased when the mesh size is refined to 17.3 million with 328 nodes/hole. It suffers a much stronger lift-off coolant flow when compared to a greater over-prediction solution solved by the mesh size of 4.10 million at a blowing ratio of 1.5.

To summarize, the comparisons between simulated film cooling effectiveness and the experimental data indicate that mesh independent film cooling effectiveness for the fan-shaped hole is more challenging than that of a cylindrical hole. Film cooling effectiveness is over-predicted at blowing ratios of 0.5. The uniformity of the film is much reduced as the blowing ratio is increased to 2.5. The results again indicate that obtaining an accurate solution of film cooling effectiveness at a higher blowing ratio is a significant challenge for RANS simulations.

4.2.3. Comparison of Lateral Film Effectiveness

The lateral distribution of film effectiveness at 5D downstream of the cooling hole (x/D=5) is utilised to investigate the mesh sensitivity of which comparisons for different blowing ratios are exhibited in Figure 9. z denotes the lateral direction. The mesh sensitivity of cylindrical and fan-shaped holes is compared. The dot-markers on the curves represent the location of a mesh node and thus its distribution signifies the mesh resolution in the lateral direction. The width of fan-shaped hole exit is around 3 times that of the cylindrical hole. Consequently, the spacing between the markers on the cylindrical hole mesh is much smaller than that of the fan-shaped hole. It should be noted that the number of nodes shown in the pictures is that around the film hole rather than those across the film.

At the lowest blowing ratio of 0.5, the solutions from the cylindrical hole (Figure 9 (a)) demonstrate good mesh convergence, whilst those from the fan-shaped hole (Figure 9 (b)) do not successfully obtain a convergent film cooling effectiveness solution. When the mesh is coarser than 28 nodes around the cylindrical hole, the high gradient of film effectiveness between the centreline and uncooled regions is not adequately resolved. The solution obtained by the mesh of 58 nodes/hole can provide good agreement except in the location of the peak where still greater fidelity if necessary. For the fan-shaped hole, there exist two peaks of film cooling effectiveness and the non-symmetric feature is also observed due to the slight crossflow in the coolant channel. Meshes coarser than 40 nodes/hole do not demonstrate this feature. Moreover, mesh independent film cooling effectiveness is not achieved for any of the meshes produced. The expansion of fan-shaped hole may enhance the anisotropic turbulent mixing between the injected coolant and hot gas. It causes a greater error with RANS modelling than the straight cylindrical hole.

For the intermediate blowing ratios, namely, 1.0 for cylindrical hole (Figure 9 (c)) and 1.5 for fan-shaped one (Figure 9 (d)), the results of both cylindrical and fan-shaped holes show an interesting phenomenon. For the former, the solutions look convergent at a relatively fine mesh of 2.94 million but those obtained by the still further refined meshes change slightly near the peak. This trend is particularly important for numerical methods. It should be noticed and needs further study. For the latter, the flow pattern obtained by the mesh size of around 1 million with 80 nodes/hole is near that by 17.3 and 72.8 million. However, that by 4.10 million with 164 nodes/hole exhibits much higher film effectiveness. Compared to the cylindrical hole exhibiting gradual mesh convergence, the fan-shaped hole demonstrates large oscillation with the mesh sizes of 0.242 million and 4.10 million. Therefore, it is difficult to declare a mesh convergence even if the curves solved by the 328 nodes/hole and 656 nodes/hole are close.

The results of the cylindrical hole at the high blowing ratio of 1.5 are shown in Figure 9 (e), where the curve of the super coarse mesh is not presented because of its great discrepancy. The film cooling effectiveness drops significantly due to the high momentum coolant flow severely lifting off. The peak obtained from the meshes with both 28 nodes/hole is much lower than those solved by the finer meshes as the reattachment process is not captured. It is found that the mesh convergence is nearly achieved in this section. For the fan-shaped hole at a high blowing ratio of 2.5 (Figure 9 (f)), the film cooling effectiveness is still comparable to that at the intermediate blowing ratio. The solution obtained by three coarsest meshes demonstrates much higher effectiveness and solved peaks are relatively smooth from the centreline. It is significantly improved as the mesh is refined to 80 nodes/hole. Meanwhile, an apparent under-prediction is observed when the mesh size is further refined to 328 nodes/hole. Consequently, again the mesh convergence of local film effectiveness is not achieved for the shaped hole.

4.3. Mesh Sensitivity of Spatial Heat Transfer

The previous comparisons demonstrate that mesh convergence of the cylindrical hole generally performs well, however, that of the fan-shaped hole is apparently challenging. Thus mesh sensitivity of the spatial heat transfer is further investigated for the cylindrical hole in this section.

4.3.1. Mesh Sensitivity of the Boundary Layer

The mesh sensitivity of the boundary layer profiles is exhibited in Figure 10. Three positions of profiles are demonstrated at locations 1D, 5D, and 10D downstream of the cooling hole. They are depicted on the top of Figure 10 (a), where the dashed line represents the centreline and three black crosses denote the positions of profiles. It should be noted that a finer mesh around the hole represents a finer nodes distribution across the boundary layer as well. The markers on each of the plotted profile curves demonstrate the mesh node distribution at the wall normal direction. The red line represents the results obtained by the coarsest mesh while that coloured black denotes those from the finest mesh.

The first column of Figures 10 (a) to (c) demonstrates the normalized velocity profile (u/U). The range starts from 0.4 to enhance the readability of the interesting sections. The second column is the normalized turbulent kinetic energy (k/K) that is normalized by the local maximum value solved by the finest mesh. Furthermore, the non-dimensional temperature defect (s) is shown in the third column. It should be noted that the current non-dimensional temperature defect is slightly different from Equation (7). Here, T_r becomes the local highest temperature, T_c denotes the lowest temperature, and both are obtained by the finest mesh. Thus $s = 0$ denotes that the hot flow is not cooled while $s = 1$ represents the best cooling effect.

The velocity profile (u/U) at blowing ratio (M) of 0.5 is demonstrated in the first column of Figure 10 (a). It is apparent that the mesh fidelity significantly affects the resulting profile. At this low blowing ratio, the velocity of the coolant is slower than that of hot gas in the boundary layer. This flow pattern is not captured by the red curve representing the solution obtained by the super coarse mesh with 6 nodes around the cooling hole. The result is improved prominently when the mesh is refined to 14 nodes/hole. The solution obtained when the mesh comprises 28 nodes around the hole is close to those by the still finer meshes at the first position located at the trailing edge of hole exit. The velocity profile obtained by the mesh of 118 nodes/hole nearly achieves the mesh independent solution. For the present simulation, that represents a mesh of 2.94 million nodes.

The corresponding turbulent kinetic energy (k/K) shown in the second column of Figure 10 (a) demonstrate the challenges associated with jet-in-crossflow simulations where there is a far greater sensitivity to the mesh size than for the velocity field. It is observed that there are two peaks of k near the trailing edge of hole exit. The second peak requires high mesh resolution near half hole diameter away from the wall such that it is more difficult to accurately capture. The super coarse mesh with 6 nodes/hole can not obtain the second peak at all. It becomes much better when the mesh is increased to 58 nodes/hole. However, different from the velocity profile, the current investigated meshes can not obtain a mesh independent solution of turbulent kinetic energy at the first location — trailing edge of the hole exit, even though the mesh size is already as large as 52.4 million. It should be noted that the mesh size for a large eddy simulation is around 10 million for this case.

The non-dimensional temperature profiles (s) is exhibited in the third column of Figure 10 (a). The super coarse mesh does not obtain a correct wall temperature at all three positions where the coolant just injects at the first one, namely the trailing edge of the cooling hole exit. The solved temperature is much higher than for the case of the other meshes. Moreover, the temperature profiles solved by the mesh sizes among 0.042 million and 2.94 million nodes show the wrong trend at the position of 5D downstream. The profile behaviour indicates that the lowest temperature is not on the wall surface. The comparison indicates that the mesh should be refined to 118 nodes/hole to obtain a mesh independent solution of temperature profile.

When the blowing ratio (M) is increased to 1.0 (shown in Figure 10 (b)) the coolant momentum is increased and thus the velocity profiles (u/U) behave differently than for the blowing ratio of 0.5. The coolant ejection evidently accelerates the boundary layer flow of the mainstream. However, the super coarse mesh does not accurately capture this profile at the first location, and the extra and coarse meshes can not do it at the second position. Unlike for the lower blowing ratio case, there is an increased challenge in achieving mesh independence for the velocity profile at all three positions. This is also the case for the profiles of turbulent kinetic energy (k/K) shown in the second column. However, if we ignore the result of the super fine mesh, the mesh converges well on the velocity solution and a mesh with 118 nodes/hole nearly achieves the independent. Similarly, the mesh refinement still significantly affects the temperature profile (s) presented in the third column — something not seen for the low blowing ratio of 0.5, even the wall temperature can not obtain a mesh independent solution.

Figure 10 (c) show the mesh sensitivity study for the blowing ratio (M) of 1.5. The wall-normal direction velocity gradient 1D downstream exhibited in the first location with u/U is enhanced significantly due to the effect of high momentum coolant ejection. Only the third location 10D downstream shows the mesh convergence. The profile solved by 118 nodes/hole is near that by 236 nodes/hole. However, the profiles of turbulent kinetic energy (k/K) shown in the second column demonstrate that the mesh does not converge at all three locations. The strong lift-off coolant is observed in the non-dimensional temperature profiles (s) of the third column. This phenomenon is captured by all the mesh sizes even if the solutions obtained by the coarse mesh differ. For the comparison of wall temperatures shown by $y/D = 0$, it is interesting to note that the solution obtained by a mesh as fine as 12.4 million with 236 nodes/hole is still significantly different from that solved by the finest mesh.

4.3.2. Mesh Sensitivity of the Flow-field

The heat transfer attributes of film cooling flow significantly influence their employment. The mesh sensitivity of non-dimensional temperature is exhibited at various blowing ratios in Figures 11 to 13 which demonstrate both iso-surfaces and contours for the cylindrical hole geometry. The solved flow-fields are compared for the seven different mesh fidelities. The right-hand side of the image demonstrates four normally oriented planes at locations, 1D, 5D, 10D, and 15D. The iso-surface demonstrated on the left-hand side represents a temperature defect of 0.75 and is coloured based on the aforementioned contour legend.

For the blowing ratio of 0.5 demonstrated in Figure 11, significant differences in the non-dimensional temperature field can be observed between the super coarse and super fine meshes. The iso-surface of temperature defect obtained by the super and extra coarse meshes is short due to the rapid rate of film decay. The normally oriented slices also exhibit this phenomenon. The bottom edge of the first slice, located at the trailing edge of film hole exit, shows a region of low temperature (signified by the green contour band). However, this region is not present in the other slices. The solution of temperature field appears improved when the mesh fidelity is increased to incorporated 14 nodes around the film hole. The blue region of the contour demonstrates the temperature is closer to that of the coolant as would be expected in the region. When compared to the coarser mesh (that with 6 nodes/hole), the downstream slices also resolve lower temperatures. The iso-surface of temperature demonstrates further improvement when the mesh is increased to 28 nodes/hole; indeed, the iso-surface appears mesh independent when 118 nodes are employed around the cooling hole. However, if we focus on the first plane at the trailing edge of the hole exit, the blue region of temperature pattern representing the coolant is still changed apparently when the mesh size is refined to 52.4 million with 472 nodes/hole.

For a blowing ratio of 1.0 demonstrated in Figure 12, the iso-surface of non-dimensional temperature still exhibits a rapid decay rate for the super coarse mesh as shown in Figure 12 (a). In this case, it is interesting to note that the decay rate is observed to be lowest when 28 nodes are utilised around the film hole as shown in Figure 12 (c) with only one low-temperature spot observed at the second slice downstream. This feature is at variance with that solved for the finer mesh. It is observed that the coolant jet lifts off at the exit of the film hole before reattaching downstream. The location of this reattachment moves upstream as the mesh is further refined. Additionally, it is observed that the iso-surface of temperature with a mesh size of 12.4 million nearly achieves a mesh independent solution. It indicates the mesh convergence at a blowing ratio of 1.0 is more difficult than that at the lower blowing ratio case of 0.5. Meanwhile, the lowest temperature at each slice moves away from the wall.

When the blowing ratio is increased to 1.5 exhibited in Figure 13, the contours demonstrate that the coolant jet lifts off. This observation, however, is not successfully captured for the super and extra meshes of which the wall is over-cooled. Rather, 28 nodes around the cooling hole are required before this lift-off is observed; however, reattachment is not observed. The iso-surface of non-dimensional temperature shows a shorter length when it is increased to the medium size mesh with 58 nodes around the cooling hole. When mesh fidelity is further increased to 236 nodes around the film hole, the flow is seen to reattach at around 5D downstream. Similar to the flow at a blowing ratio of 1.0, the lowest temperature region is away from the wall. Moreover, the shape of low-temperature region changes significantly between the coarse to fine meshes. That indicates different mixing of the coolant and hot gas flows. However, the mesh independent solution of temperature field is not achieved. Mesh refinements are still observed to result in a change to the flow pattern, even when the mesh is already as fine as 12.4M mesh nodes with 236 nodes around a hole.

To summarize, compared to the film effectiveness discussed in Section 4.2, the accuracy of spatial flow variables becomes more sensitive to the mesh size. The three-dimensional temperature field (shown in Figure 11) can not achieve a mesh independent solution even at the low blowing ratio of 0.5. It is also the case when we compare the profile of turbulent kinetic energy (exhibited in the second column of Figure 10 (a)).

4.4. Mesh Sensitivity of Laterally Averaged Effectiveness

Figure 14 presents the comparisons of laterally averaged film cooling effectiveness. Note the black markers represent the experimental data. The number shown in the images signifies the number of mesh nodes around the cooling hole. The result solved by the finest mesh is presented in a solid black line. x denotes the streamwise direction. The laterally averaged film cooling effectiveness is defined by

$$\bar{\eta} = \frac{\int_{-3D}^{3D} \eta(x,z)dz}{\int_{-3D}^{3D} dz} \tag{9}$$

where z represents the lateral direction.

The film cooling effectiveness obtained by the cylindrical hole is much lower than that by the fan-shaped hole at all three blowing ratios. The fan-shaped geometry can efficiently decrease the coolant momentum at ejection. Thus the coolant can adhere to the wall surface even when the blowing ratio is high. As shown above, the film coverage obtained by the fan-shaped hole is much greater and more uniform. The laterally averaged film cooling effectiveness obtained by cylindrical hole gradually decreases as the blowing ratio increases. In contrast, the film cooling effectiveness obtained by the fan-shaped hole gradually increases with the blowing ratio.

4.4.1. Averaged Effectiveness from the Cylindrical Hole

Figure 14 (a) demonstrates the mesh sensitivity of laterally averaged film cooling effectiveness for the cylindrical hole for a blowing ratio of 0.5. The coolant jet remains attached to the external surface at this low blowing ratio. The comparison demonstrates the attainment of mesh convergence as the nodal count is increased and settles on a line that is in close agreement with the experimental data. However, from the super coarse to coarse meshes, a large discrepancy is observable between the experimental data and CFD solutions. Indeed, in some cases the laterally averaged film cooling effectiveness even indicates jet lift-off thereby alluding to the fact that such coarse meshes may capture wrong flow features at the low blowing ratio. It is found that a mesh with 118 nodes around the film cooling hole can provide a accurate solution and the convergent effectiveness agrees well with experimental dots. However, if we focus on the region close to coolant injection, the discrepancy among three finest meshes still observable.

When the blowing ratio is increased to 1.0, the mesh convergence becomes a greater challenge as demonstrated in Figure 14 (c). The curve solved by the coarsest mesh with 6 nodes/hole is not presented due to its much higher effectiveness. The effectiveness obtained by 14 to 58 nodes/hole shows a gradual increase trend of film effectiveness after liftoff. The decay is not captured. It is also apparent that for the finer meshes (118 nodes/hole to 236 nodes/hole) the calculated laterally averaged effectiveness appears convergent. Furthermore, the numerical film cooling effectiveness is slightly under-predicted when compared to the experimental data. As indicated previously, cases involving jet lift-off provide an increased challenge for computational simulations to accurately model.

For the high blowing ratio case of 1.5, shown in Figure 14 (e), the results of the mesh study demonstrate that convergence is attained. The numerical film cooling effectiveness and exhibited trends solved by the extra coarse mesh with 14 nodes/hole are significantly different from those obtained by the finer ones. The comparison indicates that the simulated data does gradually converge to the curve obtained by the finest mesh when the number of mesh nodes is greater than 28 around the film hole with a good match to experimental data with regard to the low effectiveness.

4.4.2. Averaged Effectiveness from the Fan-shaped Hole

Figure 14 (b) shows the mesh sensitivity of laterally averaged film cooling effectiveness for the fan-shaped hole at a blowing ratio of 0.5. The decay of the averaged film cooling effectiveness obtained by both 6 nodes/hole (red) and 14 nodes/hole (green) is slower than those by the finer meshes. The results for the meshes finer than 40 nodes/hole are seen to be in close agreement with one another. Whilst the averaged effectiveness obtained by these meshes is greater than that observed in the experiments, the film decay rate is similar. Meanwhile, the comparison indicates that the laterally averaged effectiveness does exhibit good mesh convergence with a mesh independent solution achieved for the mesh with 80 nodes/hole.

For the intermediate blowing ratio of 1.5, demonstrated in Figure 14 (d), the effectiveness data obtained by the meshes with 40 nodes/hole and fewer are an over-prediction as the film coverage is wide with a slow rate of decay as demonstrated in Figure 8. As the mesh is refined to 80 nodes/hole, the solution becomes close to the experimental dots, even though the decay is still similar to that of coarser meshes. Compared to these results, the simulated averaged film cooling effectiveness by the meshes with 164 nodes/hole captures a better decay even if it is apparently over-predicted. When it is further refined to 328 nodes/hole, the CFD result is in close agreement with the experimental data and to that of 656 nodes/hole. However, the result solved by the finest mesh is slightly lower than the measurement. This trend of mesh convergence is similar to that of the cylindrical hole at the blowing ratio of 1.0.

For the high blowing ratio of 2.5, shown in Figure 14 (f), the challenge of capturing the correct decay for three coarse meshes is increased due to the stronger coolant momentum. The averaged film cooling effectiveness calculated by the meshes coarser than 80 nodes/hole shows significantly slower decay than the experimental data. The decay solved by the mesh of 4.10 million with 164 nodes/hole is in good agreement with that of the experimental curve, although the simulated curve is lower than the latter. However, the solution becomes much worse when the mesh size is increased to 17.3 million with 328 nodes/hole. Therefore, it is difficult to confirm the mesh convergence of averaged effectiveness, even though the averaged film cooling effectiveness obtained by the finest mesh of 72.8 million with 656 nodes/hole is in good agreement with that by 4.10 million.

5. Conclusions

Mesh generation is critical to the success of a CFD simulation and significantly influences the accuracy of the obtained film cooling solution. The mesh sensitivity of RANS simulations is studied by using the typical cylindrical and fan-shaped cooling holes. The multi-block structured hexahedrons with good local spacing scaling are utilised to generate a family of similar quality meshes. Seven mesh sizes are employed for each configuration with the finest mesh size 4000 times greater than that of the coarsest.

The computational results are compared with those of experimental data. The comparisons indicate that RANS solutions with a medium size mesh (58 nodes/cylindrical hole and 80 nodes/fan-shaped hole) can provide a relatively accurate result. In general, it can obtain good agreement with the experimentally observed features, especially for the laterally averaged film effectiveness. Meanwhile, the RANS simulations can predict the effect of blowing ratio change.

However, obtaining a mesh independent solution of film cooling flow is challenging for RANS modelling at certain blowing ratios and certain parameters.

- For the cylindrical hole, mesh convergence of both local and averaged film effectiveness generally performs well, however, that of the spatial heat transfer provides a challenge;

- For the fan-shaped hole, only the averaged film effectiveness converges well at the low blowing ratio of 0.5 as the local effectiveness shows observable discrepancy;

- Mesh convergence becomes more challenging at a higher blowing ratio indicating stronger coolant momentum and the subsequent greater anisotropic turbulent mixing flow with hot gas.

6. Acknowledgments

The authors would like to thank the Innovate UK and Rolls Royce funded iCORE project and Engineering and Physical Sciences Research Council (EPSRC) for sponsoring this work and the permission to publish the paper.

Appendix: Mesh Sensitivity of Lateral and Streamwise Spacing

To avoid the influence of mesh spacing at the wall normal direction, this section is additionally presented. The number of nodes at both lateral and streamwise directions is still coarsened or refined by the factor of 2 while the nodes distribution at the wall normal direction remains the same as that of the fine mesh presented in Section 4 Results and Discussion. The results of RANS simulations are presented in Figures A1, A2, and A3. The comparison indicates that the film cooling flow is significantly affected by both streamwise and lateral spacing. Comparing with the respective data in Section 4, it is found that mesh convergence with the fixed wall-normal nodes spacing behaves slightly better.

For the film effectiveness demonstrated in Figure A1, mesh convergence with a fixed wall-normal nodes spacing exhibits nearly the same as that in Section 4. It is also the case for the laterally averaged film effectiveness presented in Figure A2, with a slightly better mesh convergence at the blowing ratio of 1.0.

For the boundary layer profiles shown in Figure A3, mesh sensitivity performance is quite close to that presented in Section 4 when the blowing ratio is 0.5 and 1.0. As the blowing ratio is further increased to 1.5, except the first location x/D=1, mesh convergence becomes better for all three investigated variables (velocity, turbulent kinetic energy, and temperature).

References

[1] S. Acharya, Y. Kanani, Advances in film cooling heat transfer, in: Advances in Heat Transfer, Elsevier, 2017, pp. 91–156.

[2] J. C. Han, Fundamental gas turbine heat transfer, Journal of Thermal Science and Engineering Applications 5 (2013) 021007.

[3] T. F. Fric, A. Roshko, Vortical structure in the wake of a transverse jet, Journal of Fluid Mechanics 279 (1994) 1.

[4] L. Gevorkyan, T. Shoji, D. R. Getsinger, O. I. Smith, A. R. Karagozian, Transverse jet mixing characteristics, Journal of Fluid Mechanics 790 (2016) 237–274.

[5] B. Zang, T. H. New, Near-field dynamics of parallel twin jets in cross-flow, Physics of Fluids 29 (2017) 035103.

[6] K. Mahesh, The interaction of jets with crossflow, Annual Review of Fluid Mechanics 45 (2013) 379–407.

[7] J. Ahn, M. Schobeiri, J. C. Han, H. K. Moon, Effect of rotation on leading edge region film cooling of a gas turbine blade with three rows of film cooling holes, International Journal of Heat and Mass Transfer 50 (2007) 15–25.

[8] M. Fraas, T. Glasenapp, A. Schulz, H.-J. Bauer, Optimized inlet geometry of a laidback fan-shaped film cooling hole – experimental study of film cooling performance, International Journal of Heat and Mass Transfer 128 (2019) 980–990.

[9] L. L. Yuan, R. L. Street, J. H. Ferziger, Large-eddy simulations of a round jet in crossflow, Journal of Fluid Mechanics 379 (1999) 71–104.

[10] S. Muppidi, K. Mahesh, Direct numerical simulation of round turbulent jets in crossflow, Journal of Fluid Mechanics 574 (2007) 59.

[11] E. Sakai, T. Takahashi, H. Watanabe, Large-eddy simulation of an inclined round jet issuing into a crossflow, International Journal of Heat and Mass Transfer 69 (2014) 300–311.

[12] W. Fu, W. Chao, M. Tsubokura, C. Li, W. Wang, Direct numerical simulation of film cooling with a fan-shaped hole under low reynolds number conditions, International Journal of Heat and Mass Transfer 123 (2018) 544–560.

[13] P. Kalghatgi, S. Acharya, Flow dynamics of a film cooling jet issued from a round hole embedded in contoured crater, Journal of Turbomachinery 141 (2019) 081006.

[14] A. Burdet, R. S. Abhari, M. G. Rose, Modeling of film cooling—part II: Model for use in three-dimensional computational fluid dynamics, Journal of Turbomachinery 129 (2006) 221–231.

[15] A. Andreini, R. D. Soghe, B. Facchini, L. Mazzei, S. Colantuoni, F. Turrini, Local source based CFD modeling of effusion cooling holes: Validation and application to an actual combustor test case, Journal of Engineering for Gas Turbines and Power 136 (2013).

[16] V. K. Garg, Heat transfer on a film-cooled rotating blade using different turbulence models, International Journal of Heat and Mass Transfer 42 (1999) 789–802.

[17] A. Hoda, S. Acharya, Predictions of a film coolant jet in crossflow with different turbulence models, Journal of Turbomachinery 122 (2000) 558.

[18] A. Azzi, D. Lakehal, Perspectives in modeling film cooling of turbine blades by transcending conventional two-equation turbulence models, Journal of Turbomachinery 124 (2002) 472.

[19] J.-M. Miao, H.-K. Ching, Numerical simulation of film-cooling concave plate as coolant jet passes through two rows of holes with various orientations of coolant flow, International Journal of Heat and Mass Transfer 49 (2006) 557–574.

[20] H. Yang, H.-C. Chen, J.-C. Han, H.-K. Moon, Numerical study of film cooled rotor leading edge with tip clearance in 1-1/2 turbine stage, International Journal of Heat and Mass Transfer 51 (2008) 3066–3081.

[21] F. Montomoli, A. D'Ammaro, S. Uchida, Numerical and experimental investigation of a new film cooling geometry with high P/D ratio, International Journal of Heat and Mass Transfer 66 (2013) 366–375.

[22] H. Ma, Q. Zhang, L. He, Z. Wang, L. Wang, Cooling injection effect on a transonic squealer tip—part I: Experimental heat transfer results and CFD validation, Journal of Engineering for Gas Turbines and Power 139 (2017).

[23] X. Yan, Y. Huang, K. He, Investigations into heat transfer and film cooling effect on a squealer-winglet blade tip, International Journal of Heat and Mass Transfer 115 (2017) 955–978.

[24] Y. Li, Y. Zhang, X. Su, X. Yuan, Experimental and numerical investigations of shaped hole film cooling with the influence of endwall cross flow, International Journal of Heat and Mass Transfer 120 (2018) 42–55.

[25] Y. Jiang, L. Capone, P. Ireland, E. Romero, A detailed study of the interaction between two rows of cooling holes, Journal of Turbomachinery 140 (2018) 041008.

[26] S. Ravelli, G. Barigozzi, Numerical evaluation of heat/mass transfer analogy for leading edge showerhead film cooling on a first-stage vane, International Journal of Heat and Mass Transfer 129 (2019) 842–854.

[27] A. V. Murray, P. T. Ireland, E. Romero, Development of a steady-state experimental facility for the analysis of double-wall effusion cooling geometries, Journal of Turbomachinery 141 (2019) 041008.

[28] P. Chen, L. Wang, X. Li, J. Ren, H. Jiang, Effect of axial turbine non-axisymmetric endwall contouring on film cooling at different locations, International Journal of Heat and Mass Transfer 147 (2020) 118995.

[29] D. J. Mavriplis, J. C. Vassberg, E. N. Tinoco, M. Mani, O. P. Brodersen, B. Eisfeld, R. A. Wahls, J. H. Morrison, T. Zickuhr, D. Levy, M. Murayama, Grid quality and resolution issues from the drag prediction workshop series, Journal of Aircraft 46 (2009) 935–950.

[30] D. W. Levy, K. R. Laflin, E. N. Tinoco, J. C. Vassberg, M. Mani, B. Rider, C. L. Rumsey, R. A. Wahls, J. H. Morrison, O. P. Brodersen, S. Crippa, D. J. Mavriplis, M. Murayama, Summary of data from the fifth computational fluid dynamics drag prediction workshop, Journal of Aircraft 51 (2014) 1194–1213.

[31] E. N. Tinoco, O. P. Brodersen, S. Keye, K. R. Laflin, E. Feltrop, J. C. Vassberg, M. Mani, B. Rider, R. A. Wahls, J. H. Morrison, D. Hue, C. J. Roy, D. J. Mavriplis, M. Murayama, Summary data from the sixth AIAA CFD drag prediction workshop: CRM cases, Journal of Aircraft 55 (2018) 1352–1379.

[32] P. Kalghatgi, S. Acharya, Modal analysis of inclined film cooling jet flow, Journal of Turbomachinery 136 (2014).

[33] F. R. Menter, Two-equation eddy-viscosity turbulence models for engineering applications, AIAA Journal 32 (1994) 1598–1605.

[34] P. L. Roe, Characteristic-based schemes for the Euler equations, Annual Review of Fluid Mechanics 18 (1986) 337–365.

[35] P. Moinier, J. D. Muller, M. B. Giles, Edge-based multigrid and preconditioning for hybrid grids, AIAA Journal 40 (2002) 1954–1960.

[36] P. Moinier, M. Giles, Stability analysis of preconditioned approximations of the Euler equations on unstructured meshes, Journal of Computational Physics 178 (2002) 498–519.

[37] Y. Jiang, L. He, L. Capone, E. Romero, Investigation of steady and unsteady film-cooling using immersed mesh blocks with new conservative interface scheme, in: Volume 5C: Heat Transfer, ASME, 2016.

[38] Y. Jiang, N. Gurram, E. Romero, P. T. Ireland, L. di Mare, Computational fluid dynamics investigation of the flow of trailing edge cooling slots, Journal of Turbomachinery 141 (2019) 071002.

[39] C. Misev, N. J. Hills, Steepest descent optimisation of Runge–Kutta coefficients for second order implicit finite volume CFD codes, Journal of Computational Physics 354 (2018) 576–592.

[40] C. Saumweber, A. Schulz, Effect of geometry variations on the cooling performance of fan-shaped cooling holes, Journal of Turbomachinery 134 (2012).