EVALUATION OF THE RANS TURBULENCE MODEL PREDICTIVE CAPABILITY OF FLAT PLATE FILM COOLING

Ferdaus F1,*, N Raghukiran1

1, Centre for Automation, School of Mechanical Engineering, Vellore Institute of Technology, Chennai, India
*Corresponding author, E-mail: Fadyfithu@gmail.com

Abstract. The two-equation turbulence models used for the present study are the commonly used standard k-ε model and k-ω model. In order to achieve this target, numerical simulation was initiated in Ansys Fluent to simulate a flow over a flat test surface with a diameter of 4mm straight, circular film cooling hole at angled injections of 25°, 30°, 35° and 40°. The comparison between the numerical calculations and the theoretical results showed the standard k-ω turbulence model gave better predictions against those with the standard k-ε turbulence models. The ability of k-ω model in closely predicting the cooling behavior is due to the precise modeling of the lateral spreading of the film. The isotropic two-equation turbulence models exhibited a huge dissent. The results also indicated that increasing the mass flow rates in the mainstream channels reduces the temperature distribution along the stream-wise direction.

KEYWORDS: Film Cooling; Turbulence model, Gas turbine engine, film cooling effectiveness;

NOMENCLATURE:

Temperature of hot gas (K) \( T_g \)
Adiabatic wall temperature \( T_{sw} \)
Cold air temperature \( T_c \)
Adiabatic effectiveness \( \eta \)
Ratio of coolant to mainstream mass flux \( \text{Blowing ratio} \)
Reynolds Averaged Navier Stokes \( \text{RANS} \)
Large Eddy Simulation \( \text{LES} \)
Shear stress transport turbulence model \( \text{SST} \)
Diameter (mm) \( D \)
Turbulent kinetic energy \( (m^2/s^2) \) \( K \)
Temperature (K) \( T \)
Mainstream velocity (m/s) \( u_g \)
Streamwise direction (mm) \( x \)
Distance in wall coordinates \( y^+ \)
The coefficient of heat transfer with no film cooling \( h_0 \)
The non-dimensional resolved velocity (m/s) \( U_j \)
Spatial coordinate (mm) \( X_j \)
Dissipation rate \( \varepsilon \)
Turbulent viscosity \( \mu_t \)
Thermal conductivity (W/mK) \( \lambda \)
Mean velocity flow \( U \)
Turbulent intensity \( I \)
Turbulence model \( C_{mu} \)
Frequency of turbulence \( \omega \)
Revolutions per minute \( \text{rpm} \)

1. INTRODUCTION

Film cooling technology plays an important part to satisfy the thermal protection of turbine hot unit components. Cooler air is bled from the compressor stage and discharged through small discrete holes on blade walls, resulting in a thin cool insulating layer along the turbine blade's external surface. As cooling
air is extracted from the compressor stage for film cooling purpose, the consumption of more coolant will affect the engine's performance [1, 2]. This technique is considered an essential task as it ensures an acceptable and reasonable turbine component lifetime allowing higher turbine inlet temperatures. Since the flow in a turbine is three-dimensional and the temperature distribution is non-uniform, some locations are more vulnerable to thermal damage than others, so specific cooling performance is necessary at specific locations. To attain an enhanced cooling performance, various cooling hole arrangements have been investigated in the turbine's passage, such as film cooling on the suction and pressure side. Film cooling involves two different cooling techniques as air cooling and liquid cooling [3]. Another type of cooling the turbine was the study of new turbine blade materials that could perform under high temperatures. Among the two film cooling methods, Air cooling is considered the best cooling method as it has allowed the engines to operate at temperatures over the temperature limits of their materials, therefore creating more thrust. The film cooling technique is extensively incorporated in a gas turbine engine's combustion chambers, gas turbine blades, reheat nozzle of aircraft engine, and in rocket and ramjet nozzles [4, 5].

An extensive study has been dedicated to understanding film-cooling behaviour and its association with the mainstream flow. In general, the performance of film cooling is influenced by various parameters, such as wall curvature, injection angle, blowing ratio, mainstream turbulence, hole arrangement and configuration, mainstream and coolant Reynolds number, momentum ratios and a few more to say. Since the blades are under rotation at high speeds, extra forces are introduced, influencing the already intricate cooling process [6]. A complete and detailed review was given by Nasir, Bogard and Thole about the various factors affecting the performance of film cooling [6,7,8,9].

Over the past five decades, many researchers have extensively studied film cooling behaviour and the coolant air interaction with the mainstream flow [9]. Most of studies, the film cooling are limited to simple geometries, for example, Goldstein, surveyed the two-dimensional flat and curved surfaces incompressible flow and steady. In general, majority of the computations involved in film cooling does not go past the isotropic-two equation type[10]. The standard k-ε model was performed in numerical analysis on a flat surface at low free-stream turbulence intensity. The results shows that the near-wall treatment and isotropic turbulence modelling was unfit to seize jet lift-off. Moreover, it also highlighted that anisotropic turbulence modelling would be suitable to determine the downstream features, especially the coolant’s lateral spreading [11, 12]. Hassan and Yavuzkurt focused on comparing four separate bi-equation models to foresee film cooling performance. These turbulence models engaged in their study were the realisable k-ε model, the RNG k-ε model and the standard k-ε model. Their competencies, limitations and the relative performance were investigated and presented to predicting the film cooling effectiveness. For further improvements, the standard k-ε turbulence model was chosen as the best suitable model for the simulation of film cooling flows [13]. Azzi presented an evaluation of two turbulence models regarding their predictive efficiency in replicating near-wall heat transfer and flow physics. Their findings clearly stated that the model could exactly foresee the temperature field's span wise distribution and moderate secondary vortices' potentially by using anisotropic eddy-viscosity/diffusivity turbulence model [15, 16].

To predict a flat plate's film cooling performance by a row of laterally injected jets, Lakehal used a Navier-Stokes equation solver with a multi-block technique to compare the measured and calculated temperature and velocity fields with the standard k-ε model and the k-ε based two-layer model. The results showed that k-ε based two-layer turbulence model significantly progressed compared to the standard k-ε model with wall functions [17].

Many studies of film cooling on a flat surface are still being investigated. Film cooling on a flat surface has been the focus of study for a long time. Since the film cooling method is subjective by complicated vortical structures, the fundamental grasp of the vortex dynamics and their relation is fundamental to film cooling performance. Mcmachon, Moussa, Roshko and Fric studied the vortical structures following the transverse jet. Coelho and Kelso investigated the horseshoe vortex system. Scorcer, Greber and Kamotani also examined the counter-rotating vortex pairs. Lim studied the development and genesis of the rotating vortex
pairs and their part in the fusing process with the help of Karagozian and Cortelezzi. Bogard and Thole understood that film cooling performance is subjective to numerous parameters like turbulence intensity, angle of injection, airfoil geometry, film hole shape, Reynolds number, pressure ratio, and blowing ratio. The most important factors such as the angle of inclination and blowing ratio play a vital role in film cooling problems since they manipulate the vertical penetration and lateral spreading of the jet, thereby impacting film cooling performance [19, 20, 21].

The present work focuses on predicting the appropriateness of multiple two-equation turbulence models for film cooling performance, and the literature review provided will critically review and analyse the previous findings supporting this topic.

2. Methodology
Modern fluid dynamics is a branch of science that deals with empathising and modelling fluid flow occurrences. There are three primary methods in solving fluid flow problems. They are experimental, theoretical and computational. Although each method has its approach in solving the problems, they have the same motive to achieve the objective, i.e., to attain the best possible practical solution at low cost but with good accuracy. To fulfil the target, these three methods interrelate with each other. The present study trails the computational approach for the numerical analysis. To compare validated models with the experimental data, the study chose Ansys Fluent (CFD) to simulate the flow properties over a flat plate to obtain suitable turbulence models in forecasting the film cooling effectiveness.

The simulation carried out in the present study is performed on Ansys Fluent, a computational fluid dynamics (CFD) software. It was used to study heat and mass transfer. The CFD codes are configured, covering the numerical algorithms that can intercept the fluid flow problems. The Ansys Fluent offers numerical resolutions of partial differential equations with respect to governing airflow and transfer of heat in a discrete method. This software comprises three main sections, i.e. Pre-processor, Solver and Post-processor as shown in figure 1.

The figure 1 shows the methods taken to simulate and analyse the flat test surface with a film cooling hole. This section covers various methods and assumptions required to analyse the fluid flow in Ansys fluent.
2.1 Adiabatic Effectiveness

The load of heat conducted on an exposed surface can be calculated by the 1-D convective heat transfer equation:

$$Q_o'' = h(T_f - T_w)$$

Where $h_o$ is the coefficient of heat transfer with no film cooling, $T_{inf}$ is the mainstream temperature, and $T_w$ is the wall's temperature. As shown in the figure 2, the heat transfer on the blade's surface can be considered a two-layer model.

From the figure 2, coolant temperature $T_c$ that ejects from the film-cooling hole develops a thermal insulating layer at temperature $T_f$ on the surface of the blade, which transforms the heat load equation as:

$$Q_o'' = h(T_f - T_w)$$

Where $h$ is the coefficient of heat transfer with film cooling. Due to the mixing of hot turbulent mainstream air at a steady-state, the film temperature $T_f$ eventually increases downstream of the film cooling hole. The film cooling performance is generally labelled by the adiabatic film cooling effectiveness $\eta$, which is termed as a non-dimensional temperature. In this ongoing study, the adiabatic cooling effectiveness $\eta$ was chosen as the fundamental factor in representing film cooling performance. This parameter is defined as,

$$\eta = \frac{T_g - T_{aw}}{T_g - T_c}$$

Where $T_g$ is the mainstream temperature, $T_{aw}$ is the adiabatic wall, and $T_c$ is the coolant temperature.

2.2 Governing equations

The mathematical representation for film cooling uses 3D-time dependent Navier-Stokes equations for compressible and Newtonian fluid that comprises mass conservation, energy and momentum. The mentioned equations are non-dimensional and can be expressed as:

$$\frac{\partial U_j}{\partial x_j} = 0$$

$$\frac{\partial U_i}{\partial x_t} + \frac{\partial U_i \partial U_j}{\partial x_j} = \frac{\partial p}{\partial x_i} + \frac{1}{Re} \frac{\partial^2 U_i}{\partial x_j^2} + \frac{\partial \tau_{ij}}{\partial x_j} + f_i$$
Thus, from the above calculations, the default values of the model constants are as shown in table 1

| C1ε | C2ε | Cμ | σk | σε |
|-----|-----|----|----|----|
2.4 Standard k-\omega turbulence model

The standard k-\omega turbulence model is used to capture the effect of flow states of turbulent. This model fits the RANS category of turbulence models. This model comprises shear flow spreading, compressibility effects and low Reynolds number. They are relevant to wall-bounded flows and free shear flows. This is a resultant of more precise predictions that can be achieved for free shear flow spreading rates.

Mathematical representation of standard k-\omega turbulence model:

\[ k = \frac{3}{2} (UI)^2 \]

Where, \( k \) – turbulent kinetic energy, \( U \) – mean velocity flow, \( I \) – turbulence intensity

The turbulence intensity can be defined as:

\[ I = \frac{\dot{u}}{U} \]

\( \dot{u} \) - root mean square of turbulent velocity fluctuations and this can be defined as:

\[ \dot{u} = \frac{1}{3}(\dot{u}^2_x + \dot{u}^2_y + \dot{u}^2_z) = \frac{2}{3}k \]

\( U \) – mean velocity can be expressed as:

\[ U = \sqrt{U_x^2 + U_y^2 + U_z^2} \]

Therefore, turbulent dissipation rate can be determined by using the formula: \( \omega = C_\mu \frac{k^{\frac{1}{3}}}{\mu^\frac{1}{4}} \)

Where, \( C_\mu \) – turbulence model (0.09), \( l \) – Turbulent length scale

In this case, the turbulent viscosity is expressed as \( \nu_t \) and is calculated as: \( \nu_t = \frac{k}{\omega} \)

Assumptions: 'k' and 'c' are independently employed to capture the flows influenced by length scales variable and to simulate the velocity field. The gradient diffusion hypothesis to correlate the Reynolds stresses to turbulent viscosity and mean velocity gradients are applied by the k-\( \varepsilon \) and k-\( \omega \) models. The turbulent viscosity is expressed as the product of turbulent length scale and turbulent velocity. Since the k-\( \varepsilon \) model is based on the eddy viscosity concept, the equation becomes,

\[ \mu_{eff} = \mu + \mu_t \]

The k-\( \varepsilon \) model adopts that the turbulence viscosity is interrelated to dissipation rate and turbulent kinetic energy by the equation,

\[ \mu_t = C_\mu + \rho \frac{k^2}{\varepsilon} \]

We assume \( C_\mu \) is the model constant that is 0.09 in the present study by default. Likewise, the k-\( \omega \) model tells that the turbulence viscosity is interrelated to the and turbulence kinetic energy and turbulent frequency by the following equation,

\[ \mu_t = \rho \frac{k}{\omega} \]

Where \( \omega \) is the frequency of turbulence. The benefit of the k-\( \omega \) model is that it can be employed with a low Reynolds number model where the boundary layer is solid, and the viscous sublayer can be resolved. This model is exclusive of the intricate non-linear damping functions necessary for the k-\( \varepsilon \) model. The turbulent intensity value and turbulent length scale specification method is chosen, from the above formula can be calculated.
2.5 Computational details and model

For this study, the flat test surface design was modelled in Catia V5, and further simulation of fluid flow was carried out in Ansys Fluent. To import the Catia design into Ansys Fluent, the file was converted from the CAT part to the IGS file. For selecting suitable turbulence models, the RANS modelling strategy was used for the simulation, such as the standard k-ω and k-ε model. In order to validate, the CFD chose the same geometry with reference to Tao [5]. The flow domain examined in this present study is shown in the figure 3. From the figure 3, the coordinate system used is such that the x-axis is along the mainstream direction, the y-axis stays perpendicular to the test surface, and the z-axis is along the direction of the film-cooling hole.

![Figure 3](image)

Figure 3. (a) Coordinate system and computational model (b) Flat test surface with inclined 30-degree film cooling hole.

Figure 3a shows 1 is the mainstream inlet, 2 is the mainstream outlet, and 3 is inclined straight 30-degree film cooling hole along their respective axis. The test surface's geometry has an inclined straight 25, 30, 35 and 40 degree film cooling hole installed which is parallel to the mainstream, The film cooling-hole diameter was 4 mm.

Firstly, as shown in the figure 3b, the test surface's geometry is used to validate in this current study with the same dimension constraints mentioned above. A grid independence study is performed with respect to adiabatic effectiveness by using the formula mentioned in section 3.1, which will be further discussed in the following section.

2.6 Boundary conditions

The boundary conditions set for the follows Tao reference [5]. The mainstream gases are assumed as air with an inlet temperature of 321.15 K, and the inlet mainstream flow is considered to have a mass flow rate of 0.2kg/s. For the coolant inlet, the air is taken as a medium for gas flow with a temperature of 311.15 K and the coolant inlet flow is measured to have a mass flow rate of 1.906e-4 kg/s. Centred on the mainstream velocity and hydraulic diameter of the mainstream channel, the Reynolds number is set to be 1.45 x 10^5. The momentum ratio and averaged density ratio is set to be 0.285 and 1.026, respectively. Since the flow field is incompressible, the Mach number of the mainstream is 0.0432, and the temperature ratio is 0.968, which is the default. For both the mainstream and coolant, the turbulent intensity is assumed to be 5%. No-slip and adiabatic condition (heat flux =0) is imposed at the solid wall boundary, and static pressure, i.e., 1atm (101325 Pa), is applied at the outlet. The COUPLED scheme is selected for the pressure-velocity coupling, and for spatial discretisation, the second-order upwind is preferred for turbulent kinetic energy, momentum and turbulent dissipation rate. The y+ significance is kept within the limits of the viscous sublayer throughout the simulation. The table 2 shows the boundary conditions applied for the current study.
Table 2. Boundary conditions of the solver for the film cooling

| Sl. No | Conditions                              | Variables               |
|-------|----------------------------------------|-------------------------|
| 1     | Mass flow rate (mainstream)            | 0.3 kg/s                |
| 2     | Mass flow rate (coolant)               | $3 \times 10^{-4}$ kg/s |
| 3     | Constant Mainstream Reynolds Number    | $1.45 \times 10^5$      |
| 4     | Mach number of mainstreams             | 0.0432                  |
| 5     | Momentum ratio                         | 0.285                   |
| 6     | Density ratio                          | 1.026                   |
| 7     | Temperature (mainstream)               | 321.15 K                |
| 8     | Temperature (Coolant)                  | 311.15 K                |
| 9     | Temperature ratio                      | 0.968                   |
| 10    | Turbulence Intensity                   | 5%                      |
| 11    | Surface of the wall                    | Adiabatic Boundary condition and no slip |
| 12    | Outlet                                 | 1 atm (101,325 Pa)      |
| 13    | Convergence                            | $1 \times 10^{-3}$      |
| 14    | Iterations                             | 500                     |

3. Results & Discussion

3.1 Film cooling - Angle of deflection

In order to find the film cooling effectiveness, this paper will focus on the various deflection of angles such as $25^\circ$, $30^\circ$, $35^\circ$, and $40^\circ$. The mesh size was considered 12 mm with quad and triangle. The figure 4 shows the temperature distribution of deflection angles such as $25^\circ$ and $35^\circ$. And the figure 5 and 6 shows the temperature has distributed near the film cooling area and the gradually it has increase the temperature. If confirm that with the help of film cooling we can reduce the temperature. It is also confirm that the angle of deflection of $30^\circ$ give more cooling effectiveness compares to others. Furthermore this research will more focus on the grid independence test and turbulence models such as k-ε and k-ω.

![Figure 4. Temperature distribution with a) Angle of deflection is 25 b) Angle of deflection is 35 degree](image-url)
3.2 Grid Generation

The grid independence analysis for this present study was performed using the CFD environment "Ansys Fluent" software based on a finite volume method to elucidate the partial differential equation such as mass, momentum and energy. A grid independence study was performed in order to validate the mesh being used for this current study. A CFD solution’s reliability depends on the grid results or experimental data. The result of a coarser mesh and finer mesh can neither be the alike. To obtain good results, the mesh should be varied to get the acceptable slab of tolerance, found from the grid independence test. This test can be performed by varying the mesh size from course to fine and checking the output result from each mesh.
obtained, and when the mesh does not inhibit the result significantly, the minimum mesh size can be selected for the final solution out. The mesh used in this work is shown below in the figure 7, and it is made up of Quad/tri grids, generated using an automatic method reducing the element size.

![Figure 7. Computational mesh using Ansys Software with parameter set window](image)

The grid independence test performed in this study is executed using parametric analysis in Ansys fluent with respect to different control parameters such as velocity and temperature. To perform the parametric analysis, the method follows three steps where initially, the geometry has meshed with element size 2mm, and the respective nodes and elements are noted down as the input parameters that are to be used. Secondly, the boundary conditions are set, and simulation is made to run for 500 iterations in order to converge the residuals to obtain better accuracy. Finally, after the solution is converged, various output parameters of the desired result reports can be obtained, such as average temperature, output-Velocity (Average velocity) and maximum velocity under the surface integrals, and the results obtained can be noted down as the output parameters to check how they vary with the mesh for the element size 2mm as shown in the figure 7.

Subsequently, in the parameter set window, as shown above, various mesh element size is given such as 4mm, 6mm, 8mm, 10mm and all the design points are selected to start the calculation to obtain the results corresponding with their respective mesh element size. After all the calculations are performed, the parameter table data is exported to excel as .CSV file format where the graphs can be plotted between cell number (mesh elements) and the output parameter to check the variations.

### 3.3 Grid independence test

The grid independence test was performed with respect to section (3.4). As discussed earlier, the parameter set window with varying mesh numbers is shown in the figure 8 and different graphs were plotted between the number of mesh numbers and the output parameter.
Figure 8. Results of Grid Independence Test with respect to the element size

The figure 8 shows the mesh used, and it was made up of Quad/tri grids executed by the automatic method. The mesh generated is automatic using different grids with 45119, 56457, 87990, 166470, 871640 cell numbers.

Figure 9. a) Grid independence Test b) Temperature distribution for various mesh number

As shown in figure 9, the comparison between all the mesh sizes is based on film cooling effectiveness in the span-wise bearing. In contrast to the adiabatic effectiveness, experimental data [5], both current study and experimental data show similar agreement. From the figure 9 and experimental data, the projected percentage error was 0.75% as grid increased from 87990 to 166469, while a maximum of 1.50% was recorded as the grid amplified from 56457 to 87990. So, the mesh size of 87990 is used in the simulation to reduce the calculation time. The results obtained were performed by using the formulation of adiabatic effectiveness and percentage error analysis. The table 3 shows the corresponding output parameters with respect to their varying mesh number.

Adiabatic effectiveness ($\eta$) = \[ \frac{Tg-Taw}{Tg-Tc} \]

Percentage error = \[ \frac{Actual\ value-Expected\ value}{Actual\ value} \times 100 \]

Table 3. Parametric analysis output parameters

| SLNo | Mesh number | Adiabatic effectiveness | Average temperature (K) | Velocity-out (ms\(^{-1}\)) |
|------|-------------|-------------------------|--------------------------|-----------------------------|
| 1    | 871636      | 0.02                    | 320.9187                 | 22.66                       |
| 2    | 166469      | 0.08                    | 320.33107                | 22.02                       |
| 3    | 87990       | 0.17                    | 319.41545                | 21.07                       |
| 4    | 56457       | 0.2                     | 318.61099                | 20.24                       |
| 5    | 45119       | 0.36                    | 317.53703                | 19.14                       |

3.4 Validation on the Turbulence Models

A primary investigation has been carried out to validate the current model with the results of Tao [5]. For validation, the CFD chose the same geometry and boundary conditions mentioned in section 3.1 and 3.2, respectively. The simulation is performed in Ansys Workbench using fluent flow. The analysis is performed to validate the two-equation turbulence models.
The primary purpose of this study is to find how differently the values of an independent variable under a specific set of assumptions can influence. To perform sensitivity analysis under static conditions, the mass flow rates of mainstream inlet and mainstream coolant are set to 0.3 kg/s and 3\times10^{-4} to compare and analyse the effects on the results obtained. All other boundary conditions are set accordingly, as mentioned in table 2. The figure 10 and 11 shows the temperature contour results of the sensitivity analysis performed, and the discussion of results under static conditions are as follows;

Figure 10. Temperature contour of standard k-\(\varepsilon\) model

Figure 11. Temperature contour of standard k-\(\omega\) model

For the circumstances with a low momentum ratio, the jet film might not be much effective for jet lift-off and penetration through the mainstream and then reaffix to the downstream of the wall. Under these situations, the critical parameter affecting the effectiveness of film cooling is the lateral distribution of the coolant air. In this study, the momentum ratio is employed to be 0.285, which is very low. Also during the simulation, there is no jet lift-off observed. The figure 12 and 13 shows the velocity vectors near the film cooling hole, and it can be observed that there is no jet lift-off noted.
For comparison, the results were plotted with respect to adiabatic effectiveness and various grids, as shown in figure 14. The plot shows that the standard k-ε and standard k-ω model predicted a slow decline of adiabatic effectiveness along the stream. As shown from figure 14, the two turbulence models were observed to significantly decrease the cooling effectiveness nearby the film cooling hole. As an example for the standard k-ε model, the film cooling effectiveness starts depreciating from 0.6783 at the location near grid number 22057, and for the standard k-ω model, the cooling effectiveness value starts decreasing from 0.7019 near the grid number 177190. After this fall in effectiveness near the film cooling hole, the film cooling effectiveness showed a natural decline. In this case, the given 2 turbulence models almost corresponded near grid 91052, and this discrepancy was observed in the region right after the rapid decreasing region near grid 22057. However, the standard k-ω turbulence model was observed to be considerably better in terms of adiabatic effectiveness, they still exhibited a huge dissent in comparison with the experimental results.
Figure 14. Comparison of film cooling effectiveness

The reason for this disagreement between the numerical computation in CFD and the experimental data could be due to the anisotropy nature of the film cooling. The literature (Mayank, 2003) communicates the turbulent mingling between the mainstream and jet is 3-Dimensional and a duo of counter-rotating vortices develops downstream of film cooling hole, owing to the interference of coolant injection, and consequent of this phenomena the film cooling flow field turns out to be anisotropic. To sum it up, the standard k-ω turbulence model represented a better agreement than the standard k-ε model compared with the experimental results of Tao [5]. Moreover, since the mass flow rates of mainstream inlet and mainstream coolant were varied, the model validation results are pretty similar to the results obtained in the sensitivity analysis with only minor deviations. It is also observed that increasing the mass flow rate reduces the temperature distribution along the direction of the stream. This effect can be observed by comparing the film effectiveness graphs of analysis, as shown in figure from 4 and 14. Since, only minimal variations were observed, a justification was made to prove that when mass flow rates increase, the temperature distribution along the stream wise direction is reduced due to increased pressure inside the solid wall, making it inefficient. The justification was made by varying the mass flow rates of mainstream inlet and mainstream coolant set to be 1.2 kg/s and 0.00012 kg/s, correspondingly. The figure 15 shows the results of temperature contour for the mentioned mass flow rates and it can be clearly understood that there is minimal distribution along the stream wise direction, thus justifying the Tao results [5] as shown in figure 16.

Figure 15. Temperature contour (mass flow rate 1.2kg/s and 0.00012kg/s)
4. Conclusion

Investigation of two-equation turbulence models i.e. the standard k-ε model and standard k-ω are analysed over a flat test surface having a diameter of 4mm with an inclined injection straight circular hole of 30 degrees with the aid of Ansys Fluent. The validation of deflection and grid independency study were performed comparing the data of open published literature and further investigations were examined on static conditions and the results obtained were drawn in tabular form and graphs. The major finding of this study indicated that the standard k-ω turbulence model gave better predictions compared with the standard k-ε turbulence models due to its ability in precisely simulating the lateral spreading of the film. However, the standard k-ω turbulence model was observed to be considerably better in terms of adiabatic effectiveness, they still exhibited a huge descent in comparison with experimental results. The possible cause for this disagreement between the numerical computation in CFD and experimental results may be due to the anisotropic nature of the film cooling field. However, the two turbulence models chosen were all with the assumption of isotropy. It was also noted that increasing the mass flow rates of both mainstream inlet and mainstream coolant, reduces the temperature distribution along the stream wise direction.

Reference

[1] Ahmed, 2013. Hindawi. Film Cooling Optimization Using Numerical Computation of the Compressible Viscous Flow Equations and Simplex Algorithm, p. 25.
[2] Ferdaus F, Kumar N, Sakhiveli G, Raghukiran N. Stage reduced counter-rotating axial compressor for jet engine. International Journal for Simulation and Multidisciplinary Design Optimization. 2021;12:3.
[3] Elsayed AM, Owis FM, Abdel Rahman MM. Film cooling optimization using numerical computation of the compressible viscous flow equations and simplex algorithm. International Journal of Aerospace Engineering. 2013 Jan 1;2013.
[4] Chen AF, Li SJ, Han JC. Film cooling with forward and backward injection for cylindrical and fan-shaped holes using PSP measurement technique. InASME Turbo Expo 2014: Turbine Technical Conference and Exposition 2014. American Society of Mechanical Engineers Digital Collection.
[5] Tao Z, Zhao Z, Ding S, Xu G, Wu H. Suitability of three different two-equation turbulence models in predicting film cooling performance over a rotating blade. International Journal of Heat and Mass Transfer. 2009 Feb 1;52(5-6):1268-75.
[6] Cherrared D. Numerical simulation of film cooling a turbine blade through a row of holes. Journal of Thermal Engineering. 2017 Apr 1;13(2):1110-20.
[7] Azzi A, Lakehal D. Perspectives in Modelling Film-Cooling of Turbine Blades by Transcending Conventional Two-Equation Turbulence Models. ASME-PUBLICATIONS-HTD. 2001;369:335-48.
[8] Ferdaus F, Ganapathi R. Computational and Experimental Testing of Aircraft Bell Nozzle. International Journal of Vehicle Structures & Systems (IJVSS). 2019 Dec 1; 11(5).
[9] Garg VJ. Heat transfer on a film-cooled rotating blade using different turbulence models. International Journal of Heat and Mass Transfer. 1999 Mar 1;42(5):789-802
[10] Azzi A, Lakehal D. Perspectives in Modelling Film-Cooling of Turbine Blades by Transcending Conventional Two-Equation Turbulence Models. ASME-PUBLICATIONS-HTD. 2001; 369:335-48.
[11] Lakehal D, Theodoridis GS, Rodi W. Computation of film cooling of a flat plate by lateral injection from a row of holes. International Journal of Heat and Fluid Flow. 1998 Oct 1;19(5):418-30.
[12] Ferdaus F, Sridhar R, Sathishkumar G, Sivabalan S. Computational Analysis of Straight Nozzle. International Journal of Vehicle Structures & Systems. 2019 Apr 1;11(2):193-5.
[13] Hassan JS, Yavuzkurt S. Comparison of four different two-equation models of turbulence in predicting film cooling performance. InTurbo Expo: Power for Land, Sea, and Air 2006 Jan 1 (Vol. 4238, pp. 701-710).
[14] Lakehal D. Near-wall modeling of turbulent convective heat transport in film cooling of turbine blades with the aid of direct numerical simulation data. J. Turbomach.. 2002 Jul 1;124(3):485-98.
[15] Lakehal D, Theodoridis GS, Rodi W. Three-dimensional flow and heat transfer calculations of film cooling at the leading edge of a symmetrical turbine blade model. International journal of heat and fluid flow. 2001 Apr 1;22(2):113-22.
[16] [19] Garg VK. Adiabatic effectiveness and heat transfer coefficient on a film-cooled rotating blade. Numerical Heat Transfer, Part A Applications. 1997 Dec 1;32(8):811-30
[17] Tyagi M, Acharya S. Large eddy simulation of film cooling flow from an inclined cylindrical jet. J. Turbomach.. 2003 Oct 1;125(4):734-42
[18] Al-Zurfi N, Nasser A. LES of rotating film-cooling performance in a 1-1/2 turbine stage. Propulsion and Power Research. 2019 Jun 1;8(2):85-107.
[21] Nadalina Jafabadi H. Investigation of Film Cooling Strategies CFD versus Experiments-Potential for Using Reduced Models.
[22] Al-Zurfi N, Nasser A. LES of rotating film-cooling performance in a 1-1/2 turbine stage. Propulsion and Power Research. 2019 Jun 1;8(2):85-107.
[23] Alzurfi N, Turan A, Nasser A, Alhusseny A. Numerical simulation of film cooling effectiveness in a rotating blade at high blowing ratios.
[24] Zhu J, Wang K, Nie C. Computational research on film cooling under rotating frame by an anisotropic turbulence model. Propulsion and Power Research. 2013 Mar 1;2(1):20-9.
[25] Nemdili F, Azzi A, Jubran BA. Numerical investigation of the influence of a hole imperfection on film cooling effectiveness. International Journal of Numerical Methods for Heat & Fluid Flow. 2011 Jan 11.