This study investigates the film cooling process on a flat plate. The task addressed relates to the identification of the most universal CFD model for different film cooling hole shapes.

The task has been solved by selecting an optimal computational mesh and determining the most suitable turbulence models for predicting film cooling effectiveness over a wide range of parameters. To investigate mesh-independence effects, four levels of polyhedral computational grids were generated. It was shown that the mesh with 5.8 million elements, selected based on the mesh-convergence analysis, performs nearly as well as a block-structured mesh with identical settings.

A validated CFD model based on a polyhedral mesh was built. A distinctive feature of the results is that the CFD model covers 4 hole geometries spaced 5D apart and inclined at 30° to the mainstream flow (classical cylindrical, fan-shaped, oval, and a diffused slot).

The results include a comparison of seven RANS turbulence models with experimental data. It was found that for the considered flow and geometric conditions, the most robust and generally applicable model is the k-ε Realizable turbulence model. Its advantages may be explained by its improved stability and better sensitivity to regions of complex flow kinematics, which enables more accurate prediction of expanding (fan-shaped and diffuser-type) holes.

Additionally, the model feasibility was verified for the 7-7-7 hole configuration. For this type of hole, a preliminary analysis of the influence of thermal barrier coating configurations on film cooling effectiveness is presented. The proposed computational model could be used for optimizing hole geometry and blowing conditions in gas turbine blade cooling applications

Keywords: film cooling, cooling efficiency, flat plate, numerical modeling, turbulence model

UDC 621.438.016.4

DOI: 10.15587/1729-4061.2026.356502

DEVELOPMENT OF A VALIDATED COMPUTATIONAL FLUID DYNAMICS MODEL FOR FILM COOLING EFFICIENCY CALCULATION

Oleg Shevchuk
Corresponding author
PhD Student, Lead Engineer
Department of Turbine
JSC Ivchenko-Progress
Ivanova str., 2, Zaporizhzhia, Ukraine, 69068
Department of Turbine Construction*
E-mail: Oleh.Shevchuk@ieee.khpi.edu.ua
ORCID: https://orcid.org/0000-0003-1837-6287

Oleksandr Tarasov
Doctor of Technical Sciences, Professor
Department of Turbine Construction*
ORCID: https://orcid.org/0000-0002-5952-3258

*National Technical University
"Kharkiv Polytechnic Institute"
Kyrpychova str., 2, Kharkiv, Ukraine, 61002

Received 12.01.2026
Received in revised form 16.03.2026
Accepted 27.03.2026
Published 30.04.2026

How to Cite: *Shevchuk, O., Tarasov, O. (2026). Development of a validated computational fluid dynamics model for film cooling efficiency calculation. Eastern-European Journal of Enterprise Technologies, 2 (1 (140)), 78–90. https://doi.org/10.15587/1729-4061.2026.356502*

1. Introduction

Over the last two decades, Computational Fluid Dynamics (CFD) simulation has undergone great evolution. CFD problems, which are based on the Reynolds-averaged Navier-Stokes (RANS) equations, can help the design engineer choose the optimal variant of a particular design, gaining knowledge about the nature of the flow in the studied area. Turbulence models offered in such commercial products as ANSYS CFX (USA), ANSYS FLUENT (USA), CADENCE FIDELITY (USA), STAR-CCM (USA), etc. have a wide range of sub-models and settings, which, together with the settings of the computational grid, can give different results. However, in order to claim that the result fully corresponds to reality, according to [1]:

a) the mathematical model must cover all aspects of the real world;

b) the numerical method must accurately solve the equations in the mathematical model.

These two conditions cannot be fully satisfied, so the numerical solution is only an approximation to reality. In the case of film cooling of turbine blades of a gas turbine engine, this almost always means that the mathematical model and numerical method must reflect the distribution of film cooling efficiency (1) along the blade perforation hole with reasonable accuracy and similarity to reality

$$\eta = \left(T_g - T_f\right) / \left(T_g - T_c\right), \quad (1)$$

where T_g is the gas temperature, T_f is the film temperature, which is a mixture of gas and cooling liquid, that is, air with temperature T_c.

Film cooling involves releasing the coolant on the surface of the gas turbine blades in order to push hot gas from them, the current level of which has reached 2000 K today and exceeds the operating temperature of existing materials. According to [2], the efficiency of film cooling is influenced by a number of factors, such as the location of the perforation holes relative to the gas flow (angles of inclination of the hole axis to the wall α and to the direction of the main flow β), the shape and density of the perforation holes of the film cooling, the number of rows, and the distance between the rows of holes, the relative length of the holes, the curvature and roughness of the surface, the presence of a thermal barrier coating (TBC), and the blowing ratio m of the film cooling hole

$$m = \rho_c w_c / \rho_g w_g, \qquad (2)$$

where ρ_c and ρ_g are the densities of air and gas, w_c and w_g are the velocities of air and gas.

The mathematical model of film cooling must take into account all of the above factors. But even then, general recommendations for choosing turbulence models or the size of the computational grid in CFD simulation cannot provide guaranteed accuracy of calculations. That is why the CFD model must be validated by comparing it with experimental studies. The validated CFD model can be further used to optimize the geometry of holes and injection modes in the problems of cooling gas turbine engine blades. This also creates prerequisites for using this model in the development of new promising types of holes that can be produced only through additive 3D printing technologies. Another important task that is planned to be solved using this CFD model in the future is the study of the effect of partial blocking of blade holes by a thermal barrier coating, which requires a sufficiently fine grid. Therefore, studies aimed at validating a computational model are mandatory and relevant.

Further development of the topic involves the construction of an expanded database of CFD calculations for various types of film cooling holes, the development of new generalized dimensionless criteria, and the formation of refined 1D correlations. That is why a validated CFD model should have a fairly wide range of application of geometric and operating parameters.

2. Literature review and problem statement

According to [3], in practical applications, all film cooling holes have either a round or so-called profiled shape. In fact, all shaped holes consist of a round throttle channel and a uniformly and symmetrically expanded (up to 15 degrees) outlet section on the hot gas surface. It is shown that the manufacturing technology of such holes is well-developed, and the adiabatic efficiency is high. It is also noted that the greatest advantage of diffuser holes is their reliability and stability of characteristics. But issues related to the cost of manufacturing and further improvement of adiabatic efficiency remain unresolved.

That is why a significant amount of work is devoted to optimizing shaped holes and finding options for increasing their adiabatic efficiency by solving CFD problems. Thus, in the review paper [4], an analysis of current (2019–2025) numerous 3D CFD studies in the field of film cooling on a flat plate, including for round and shaped holes, is carried out. It is shown that the vast majority of works use RANS (Reynolds-Averaged Navier-Stokes) methods. The reason is an acceptable ratio of accuracy and computational costs.

In work [5] it is noted that RANS methods have insufficient ability to reproduce turbulence in the interaction of a jet with a cross flow and do not have a complete coincidence with experimental data. But at the same time, they are able to predict the trend of change in the adiabatic efficiency of film cooling quite accurately, and the research approach and calculation results when using them are reliable.

An option for improving the prediction of complex phenomena, including the interaction of a jet with the main flow, is the use of Large Eddy Simulation (LES) methods. This approach was used in work [6] to study the influence of surface roughness of a shaped hole, and in [7] – to study the influence of the Reynolds number of a shaped hole located in a trench. In both cases, LES investigated a 7-7-7 fan-shaped hole, in which two lateral and one forward expansion angles are each 7°, which explains its name. However, LES methods require significantly higher computational costs with high mesh requirements, which limits their application in industrial practice.

Another option for improving the prediction of adiabatic efficiency is to use adapted turbulence models. This is the approach used in [8] for validating the CFD model of film cooling and (partially) in [9] for optimizing the trench shape. In both cases, the k-ω group models for cylindrical hole shapes on a flat plate were adapted. However, changing the coefficients often improves the result only for a specific problem, which significantly limits the validation of the adapted model compared to the standard turbulence model. In addition to the loss of universality, there is a risk of violating the physical justification and worsening the predictive ability for new shapes of film cooling holes.

In [10], the influence of the turbulent Prandtl number on the accuracy of numerical prediction of film cooling efficiency is investigated. The authors note that most CFD calculations use a constant value of the Prandtl number, which is equal to 0.85. However, in real film cooling flows, this parameter varies depending on the turbulence structure, especially in the mixing zone of the cooling air jet with the main flow. Because of this, conventional models often incorrectly reproduce the distribution of cooling efficiency, overestimating it in the center of the jet and underestimating it at the edges from the hole. The authors also showed that the real Prandtl number can vary in the range of 0.4–0.85 and this can significantly affect the predicted film efficiency. Based on the analysis of the results, the authors proposed their model, in which the turbulent Prandtl number varies depending on the transverse position relative to the jet axis. However, such a model is semi-empirical in nature and is tied to specific experimental conditions, which also limits its universality. It should also be noted that the validation in [10] was carried out only for experiments with round holes. This creates a risk that the model proposed by the authors may work worse for other types of holes.

Indeed, our literature review has shown that the vast majority of validation studies on computational models are limited to a single hole configuration or a limited number of geometries. Although this approach allows one to test the model's ability to reproduce specific experimental data, it cannot guarantee correct predictions for other hole shapes.

The literature review also showed a significant uncertainty in the selection of turbulence models for the calculation of cylindrical, profiled holes, and holes with injection into various types of recesses. Thus, in [4] it is shown that two standard RANS turbulence models are found in the works as selected based on the results of validation of the computational model, most often. These are the two-parameter RANS models k-ε realizable, proposed in [11], and k-ω SST (Shear Stress Transport), proposed in [12]. Among current papers on modeling the efficiency of film cooling on a flat surface reviewed in [4], these two turbulence models together account for about 70% of studies. Moreover, the above turbulence models are equally used for both cylindrical and profiled holes.

In addition, in [4] it is shown that structured block grids with regular topology are the most common among other types of computational grids in the study of the efficiency of film cooling on a flat surface. There is a widespread belief that the most accurate and most efficient solutions are obtained on "good" grids, similar to structured Cartesian grids or grids composed of identical ideal elements (ideal triangles, tetrahedra, etc.). This idea contradicts the modern practice of computational fluid dynamics (CFD), in which accurate solutions are calculated on practical grids, which many consider unacceptable in many geometric indicators of mesh quality [13].

When calculating film cooling from shaped holes on a flat plate, polyhedral computational grids are gaining popularity. Due to the larger number of faces, this type of grid can reduce numerical errors when transferring data between elements. This approach was used in [14], in which a polyhedral grid is used to optimize the basic profile of the aforementioned 7-7-7 fan-shaped hole. As a result of the optimization, the authors obtained holes with a complex shape with dune-shaped protruding surfaces on both sides of the hole that affect the main flow, forming a pair of vortices rotating in opposite directions. These vortices complicate the separation of the coolant from the outer surface and at the same time increase the spread of the jet. The effect of the cross-flow velocity on the cooling efficiency of a 7-7-7 fan-shaped hole using a polyhedral grid is investigated in [15].

Each work on calculating the efficiency of film cooling on a flat surface must be generalized by many parameters to form recommendations. Therefore, the preliminary validation of the calculation model is an integral part of this work and cannot be solved by general recommendations by other authors. It should also be noted that in the available literature on film cooling, there is variability in the level of detail of the criteria used in CFD calculations and analysis of the obtained data. In addition, the geometry and size of the model, as well as the selected turbulence model, must take into account the further work that the author plans to carry out.

All this allows us to state that it is advisable to conduct a study aimed at finding the most universal CFD model in terms of regime and geometric parameters for the work planned by the engineer. Substantiation of the selected CFD model for further calculations is a logical step in ensuring the reliability of the results.

3. The aim and objectives of the study

The aim of our work is to devise the most versatile CFD model of film cooling of a gas turbine blade to determine the cooling efficiency of both cylindrical and shaped holes. This will make it possible to increase the accuracy of calculations when optimizing the geometry of holes and injection modes in the problems of cooling gas turbine engine blades.

To achieve the goal, the following tasks were set:

– to investigate the influence of mesh convergence of an unstructured mesh;

– to compare an unstructured mesh with a structured block mesh with identical settings;

– to compare the results with experimental data from different turbulence models;

– to analyze the ability of the validated model to investigate the technological features of applying an TBC.

4. The study materials and methods

The object of our study is the process of film cooling on a flat plate, taking into account the specificity of the development of a cooling film on gas turbine blades, such as clogging of holes, the presence of a thermal barrier coating. Therefore, the task was to find the most universal CFD model, taking into account different shapes of film cooling holes and possible clogging of holes. Calculations were performed in the commercial three-dimensional CFD solver ANSYS FLUENT 2024 R2, which is based on pressure (pressure-based solver).

The principal hypothesis assumes that a 3D CFD model could be built that adequately describes the main flow mechanisms. The difference between the results of numerical modeling and the experiment regarding the cooling efficiency would be statistically insignificant and would be within the error of experimental measurements and numerical methods.

For the validation of the CFD model of the film cooling, an experiment was selected that was reported in [16] by Takasago Research and Development Center, Mitsubishi Heavy Industries Ltd (Japan) in 1999. This rare work is available only in the proceedings of the IGTC-99 conference but has several advantages. The cooling efficiency was studied for four different hole shapes over a fairly wide range of injection parameters and relative lengths behind the hole (from 0 to $X / D = 70 - 112$). The study of the influence of the main flow turbulence on the cooling efficiency, as well as the distribution pattern of the efficiency on the transverse planes YZ along the length X / D, is presented.

The velocity of the main flow in the experiment was 20 m/s, and its temperature was 12°C. The absolute pressure in the main flow channel was 1.025 atmospheres. The temperature of the air supplied to the holes was 42°C. An experimental study of the cooling efficiency on a flat plate was carried out for four models of rows of holes (Fig. 1). Model 1 is a standard cylindrical row of holes, located at an angle $\alpha = 30°$ and with a pitch of 5D. The diameter D in the experiment was 10 mm, and the wall thickness L_W, in which the hole is located, was 3D. Model 2 has the same parameters as model 1, but the cylindrical part of the hole at the outlet expands at an angle of 15° in both directions and forms a complex fan-shaped hole. Models 3 and 4 have the same oval shape of the outlet holes, but model 3 has an oval shape along the entire length of the hole, and model 4 has an extension from a cylindrical shape to an oval at an angle of 15°. Models 3 and 4, unlike the first two models, have an inlet part of the hole (height H in the case of model 3 and diameter D in the case of model 4) of 6.25 mm (that is, the wall thickness is 4.8D, and the hole pitch is 8D). The ratio of the outlet hole width W to the hole pitch P for models 3 and 4 is 0.6. All 4 models have a zero angle β, that is the coolant exits in the direction of the main flow.

The main flow turbulence intensity was 0.5%, and with the turbulent grid installed, the turbulence was 10%. The effect of main flow turbulence on the cooling efficiency was determined for model 2.

The experiments were carried out on a flat plate with low thermal conductivity. The adiabatic wall temperatures were measured using 0.1 mm diameter chromel-alumel thermocouples embedded in the surface. In the paper, the cooling efficiency is represented as plots depending on the parameter X / MS. Here, X is the longitudinal length from the hole downstream, M is the blowing ratio, S is the width of the equivalent gap, which is the ratio of the hole area to the pitch between the holes.

To perform the tasks set, a geometric model of film cooling was constructed, which is shown in Fig. 2. For simplicity, the model contains 1 film cooling hole with periodicity conditions. According to the experiment in [16], the film cooling hole is located at angles $\alpha = 30°$, $\beta = 0°$. The hole wall thickness L_W is 3D ($L / D = 6$), and the pitch between the cylindrical holes is 5D. The length of the model downstream from the opening is 70D (models 1 and 2) and 112D (models 3 and 4).

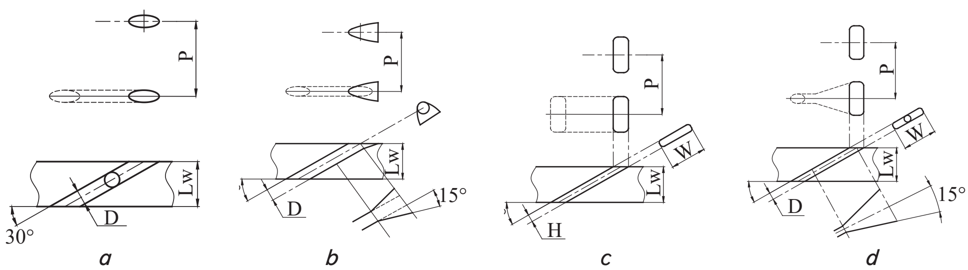


Fig. 1. The shapes of film cooling holes were investigated according to [16]:
a — model 1 (standard round holes); *b* — model 2 (shaped holes); *c* — model 3 (quasi-two-dimensional slots or oval holes); *d* — model 4 (diffuse two-dimensional slots)

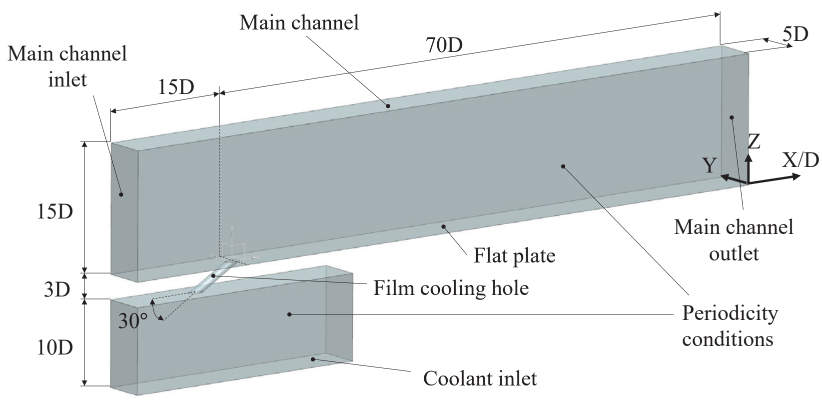


Fig. 2. Geometric model of film cooling with a standard circular film cooling hole with a diameter of $D = 10$ mm

The boundary conditions were the velocity at the inlet to the main channel $V_\infty = 20$ m/s and the turbulence intensity $Tu_\infty = 0.5\%$, as well as the physical flow rate of the coolant, which provides the required blowing ratio. The temperatures at the inlets of the conditional coolant and conditional gas were 315 K and 285 K, respectively (inverted heat flow scheme). The density ratio of the coolant and the main flow (*DR*) in this experiment was 0.9. The working environment in this study is air, which is considered as an ideal gas. The determination of the dynamic viscosity as a function of temperature was performed according to the Sutherland equation.

The Reynolds number of the main flow $\mathrm{Re} = \rho v D_h / \mu \approx \approx 105000$, where ρ and v are the density and flow velocity, D_h is the hydraulic diameter of the channel, and μ is the dynamic viscosity. The calculation of the Reynolds number in a cylindrical hole, where the diameter of the hole is taken as the characteristic dimension, namely 10 mm, indicates that the Reynolds number in the film cooling hole is approximately 12000 for the injection parameter $m = 1$.

The controlling parameter for the convergence of the calculation, in addition to the residuals, was the average temperature of the flat plate over the plane, that is, the change in the total surface temperature between iterations should not exceed 0.01°C. To calculate the blowing ratio, the weighted averaged velocity and density of the coolant on the cylindrical part of the hole were used. The averaging of the cooling efficiency on the flat plate (lateral efficiency) was performed at 26 points in the lateral direction (with a step of 1 mm) along each X/D coordinate.

5. Results of determining the most universal CFD model

5. 1. Results of the study of mesh convergence

Increasing the number of elements of the computational mesh, on the one hand, allows for a more detailed description of the computational model and the processes occurring in it, and on the other hand, increases the computational cost. It is precisely to find a sufficiently fine mesh, the further thickening of which does not give significant changes in the results, that mesh convergence analysis is performed.

To assess the absence of influence on the results of the size of the selected computational mesh (Fig. 3), 4 variants of polyhedral meshes were constructed and tested in ANSYS FLUENT MESHER 2024 R2, the characteristics of which are given in Table 1.

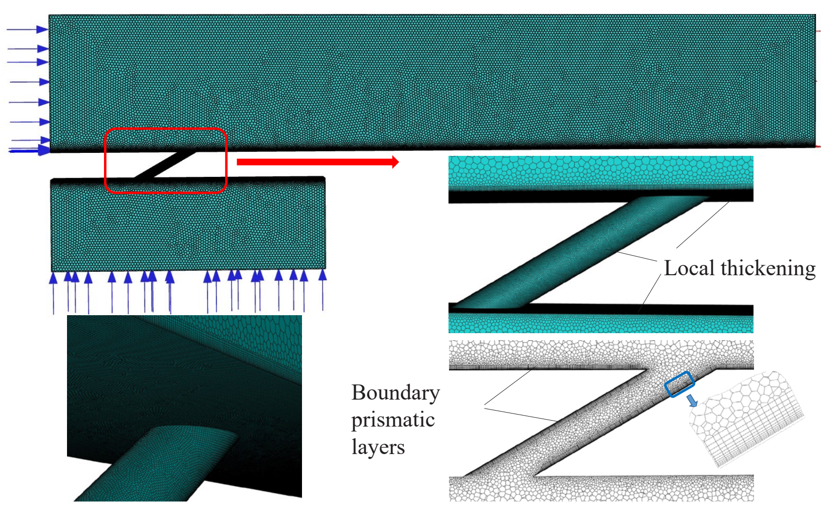


Fig. 3. Polyhedral computational mesh (3rd level — 5.8 million elements)

Table 1

Characteristics of a polyhedral mesh in the study of mesh convergence

Mesh level	Number of elements, mln.	Number of nodes, mln.	Max. element	Local thickening (Fig. 3)	Boundary prismatic layers (Fig. 3)	Minimal element of the prismatic layer	Averaged Y^+
1 ($Y^+ < 4$)	1.8	6.9	0.3D	0.075D	$N = 10$ ($k = 1.20$)	0.01D	3.3
2 ($Y^+ < 3$)	2.9	10.5		0.056D		0.0075D	2.4
3 ($Y^+ < 2$)	5.8	20.2		0.038D		0.005D	1.6
4 ($Y^+ < 1$)	9.3	27.2			$N = 20$ ($k = 1.15$)	0.0025D	0.8

The size of the maximum element was the same for all meshes and was 0.3D (3 mm). The coarse (level 1) mesh was chosen so that the Y^+ parameter on the adiabatic flat surface was less than 4, and the finest (level 4) mesh had an average Y^+ parameter less than 1 (Fig. 4). Local sizing for each mesh was performed the same for three surfaces, namely for the flat plate, the film cooling hole, and the upper part of the coolant cavity, where the entrance to the hole is located. However, the boundary wall layers for each mesh were applied the same for only two surfaces, namely for the flat plate and the film cooling hole. The boundary prismatic layers at all mesh levels are constructed in such a way that there is a smooth, rather than a jump-like, transition between the last prismatic layer and the next element, which limits the interpolation error.

The independence of the solution from the grid size was tested using the $k\varepsilon$-realizable turbulence model with an enhanced wall function. The $k\varepsilon$-realizable turbulence model with an enhanced wall function showed excellent convergence of the lateral cooling efficiency already for a coarse (level 1) grid (Fig. 5, *a*). The GCI (grid convergence index [17]) when moving to the third-level grid, calculated from the total cooling efficiency along the number of holes for the $k\varepsilon$-realizable turbulence model is $\mathrm{GCI}^{21}_{fine} = 0.55\%$. This is quite suitable for detailed studies (GCI ≤ 5%).

Choosing a coarse (level 1) grid would reduce the grid size, but this choice would be wrong. With relatively constant lateral cooling efficiency, a rather significant change in the cooling efficiency along the centerline is observed for different grids (Fig. 5, *b*). Neglecting this criterion can affect the correctness of the idea of the physics of the process. Additional calculations using turbulence models of the $k\omega$ group, namely $k\omega$ – SST and $k\omega$ – GEKO, confirm the influence of the grid on the cooling efficiency along the centerline.

The transition from the 3rd level grid to the fine grid of the 4th level has the least impact on this additional criterion for both groups ($k\varepsilon$ and $k\omega$) of turbulence models, and the nature of convergence in both cases is oscillatory (oscillatory convergence). Also, the 3rd level grid (Table 1) is acceptable for describing the presence of a thermal barrier coating on the working blade. Thus, the thickness of the thermal barrier coating on the working blade of a turbine usually does not exceed 0.4D, which is almost half the optimal value of the trench depth. Therefore, a sufficiently fine grid should be used to describe such a trench.

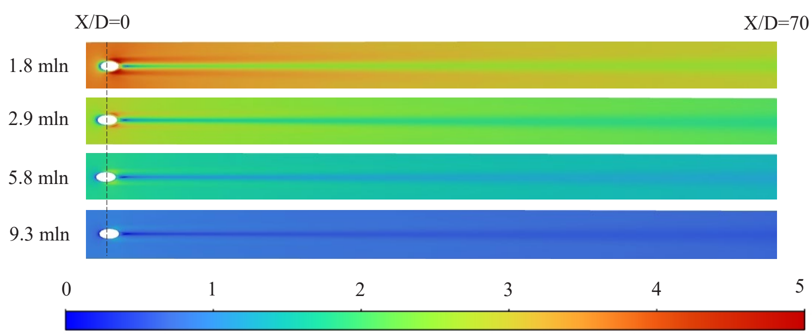


Fig. 4. Distribution of parameter Y^+ on a flat plate

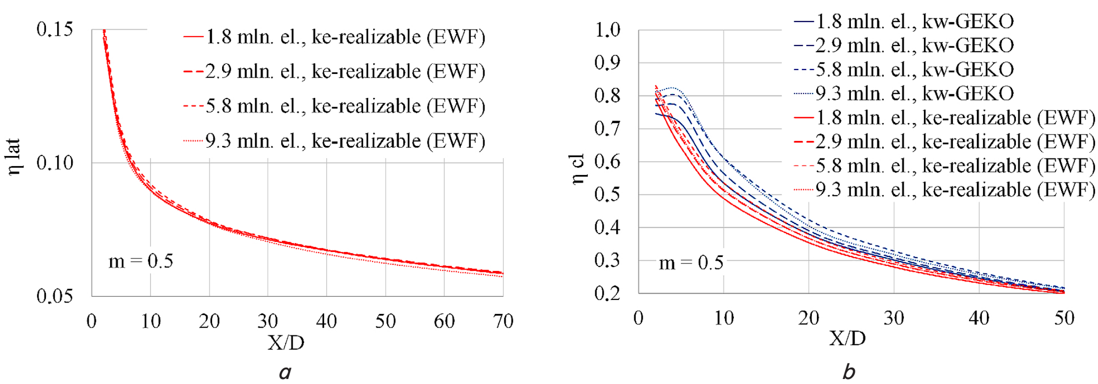


Fig. 5. Investigation of mesh convergence at the blowing ratio $m = 0.5$ with a cylindrical film cooling hole: *a* – lateral; *b* – along the center line

Thus, for further calculations, an intermediate third-level grid with a size of 5.8 million elements was selected.

5. 2. Comparison of unstructured mesh with structured block mesh

In addition to polyhedral meshes, structured meshes with the same maximum and minimum elements as the polyhedral meshes given in Table 1 were constructed in ICEM CFD 2024 R2. The conclusions about the convergence of the solution process for these meshes turned out to be close to the conclusions for polyhedral meshes, so a third-level mesh with a size of 6.2 million elements was also chosen for comparison (Fig. 6).

Fig. 7 shows a comparison between the calculations of two different types of meshes (but with the same settings) for the *k-ε* realizable turbulence model, which is more stable for each type of mesh.

The comparison showed almost identical values of lateral efficiency for the two types of grids over the entire X/D range. An additional comparison along the center line indicates similar trends of the curves of the two types of grids, but the curve of the block grid lies lower at $m = 0.5$, which is actually closer to the experimental data [18]. However, at the injection parameter $m = 1.0$, no significant difference between the two types of grids was observed. Such existing minor advantages of the block grid are not decisive, and it was decided that in further work a simpler polyhedral grid will be used. It is important that the geometric and regime conditions of the experiment in [18] are close to the experiment in [16]. Therefore, such a comparison is appropriate.

5. 3. Comparison of turbulence models in their suitability for film cooling analysis

Testing of turbulence models was performed for four types of film cooling holes (Fig. 1) and experimental conditions [16], with blowing ratios corresponding to the efficiency values of each hole close to the maximum.

Seven RANS turbulence models were applied for each hole model, including two-parameter *k-ω* SST, *k-ω* SST (Low Reynolds corrections), *k-ω* GEKO, *k-ω* WJ-BSL-EARSM, *k-ε* realizable, *k-ε* RNG, and one-parameter Spalart-Allmaras model. The presence of the *k-ω* SST turbulence model with the "Low Reynolds corrections" option in the list is explained by the low Reynolds number in the film cooling hole itself, which is about 12,000 at the injection parameter $m = 1$. The *k-ε* group models for solving the turbulent boundary layer were used exclusively with the "Enhanced Wall Treatment" option. The function includes the "Enhanced Wall Function" (EWF) and allows keeping the Y^+ parameter both less than 1 and at the coarse mesh level. The standard *k-ω* and *k-ε* models were also used in the calculations but showed the worst convergence of the calculation in their groups; therefore, they were not presented in further materials.

Fig. 8, *a* shows the results of the calculation of the lateral efficiency of model 1, which is a cylindrical hole, the efficiency of which is the highest at low blowing ratios. It can be seen that all turbulence models, except for the *k-ω* WJ-BSL-EARSM, which is much lower, are quite close to the experiment and describe the trend well. The results of the *k-ω* GEKO, *k-ω* SST and even Spalart-Allmares turbulence models are closest to the baseline experiment.

Fig. 8, *b* shows the results of the calculation of the efficiency along the centerline for the same cylindrical hole at the blowing ratio $m = 0.5$. Due to the fact that the baseline experiment does not provide an analysis of the efficiency along the centerline, a comparison is made with the experiment [18], which was carried out at different injection parameters and *DR* values. In this case, an experiment with values as close as possible to the base experiment is presented, namely at $m = 0.515$ and $DR = 0.96$.

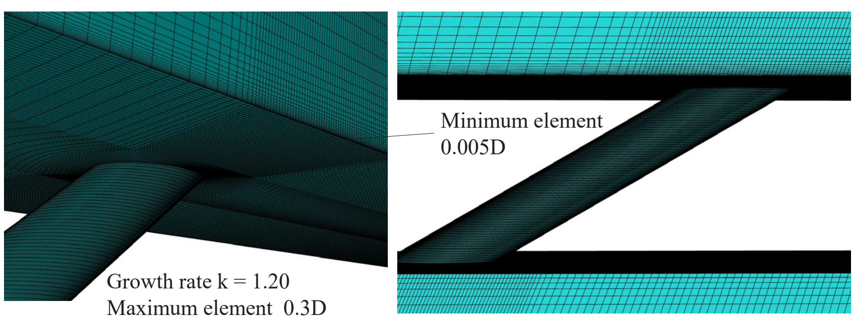


Fig. 6. Fragment of a structured computational grid (level 3 — 6.2 million elements)

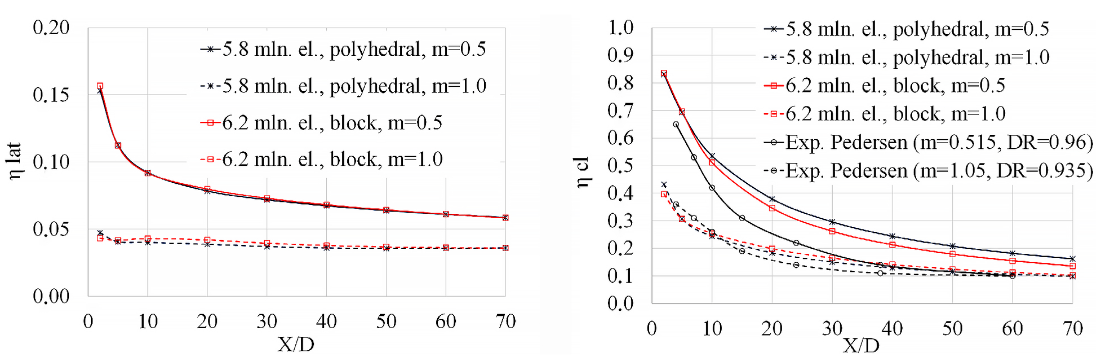
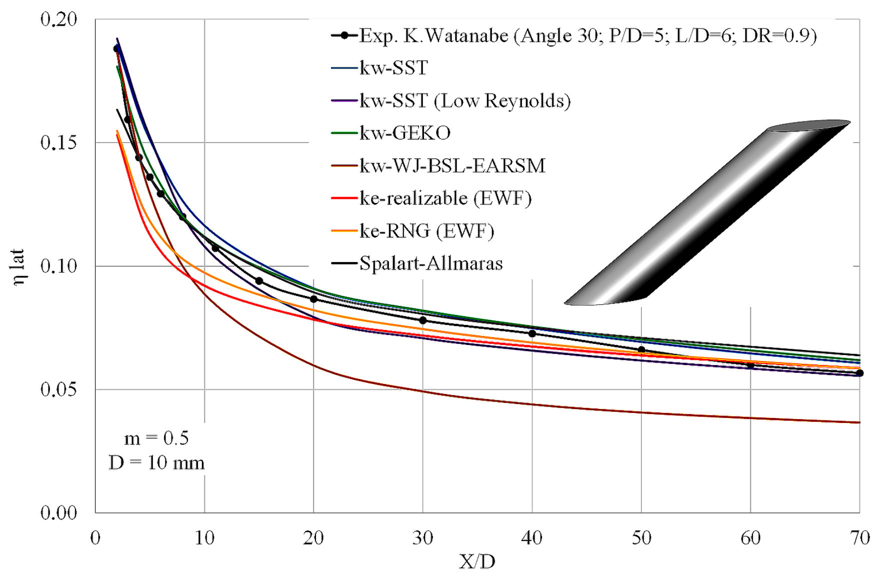
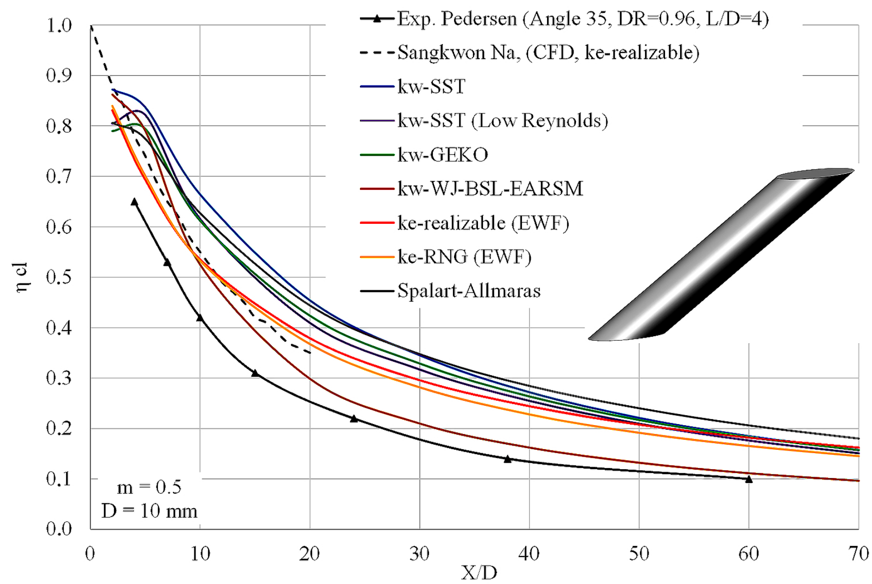


Fig. 7. Comparison of the results of the *k-ε* realizable turbulence model for polyhedral and block structured third-level meshes with a cylindrical film cooling hole at blowing ratios $m = 0.5$ and $m = 1.0$: *a* — lateral; *b* — along the centerline



a



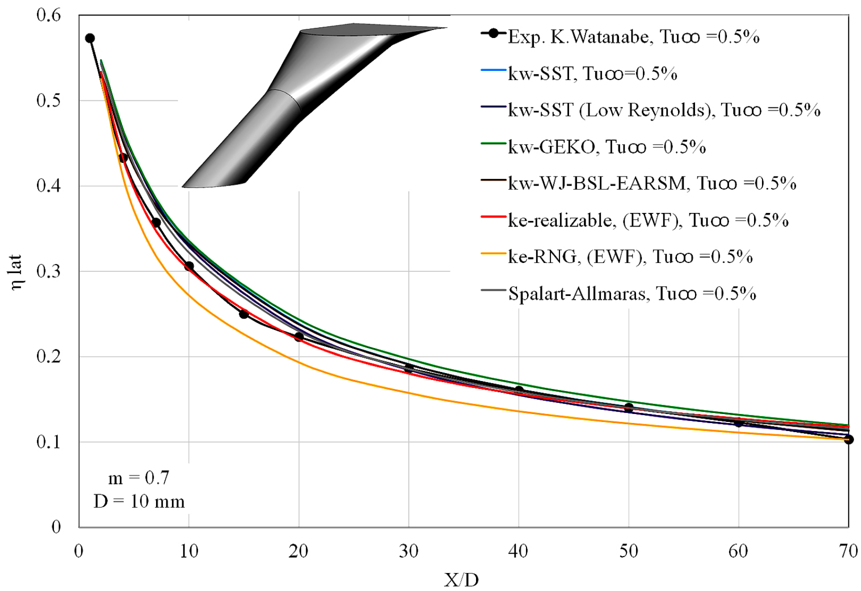
b

Fig. 8. Film cooling efficiency calculations on a level 3 grid: *a* – comparison of lateral efficiency calculations for model 1 (cylindrical hole) at $m = 0.5$; *b* – comparison of efficiency calculations along the centerline of model 1 (cylindrical hole) at $m = 0.5$

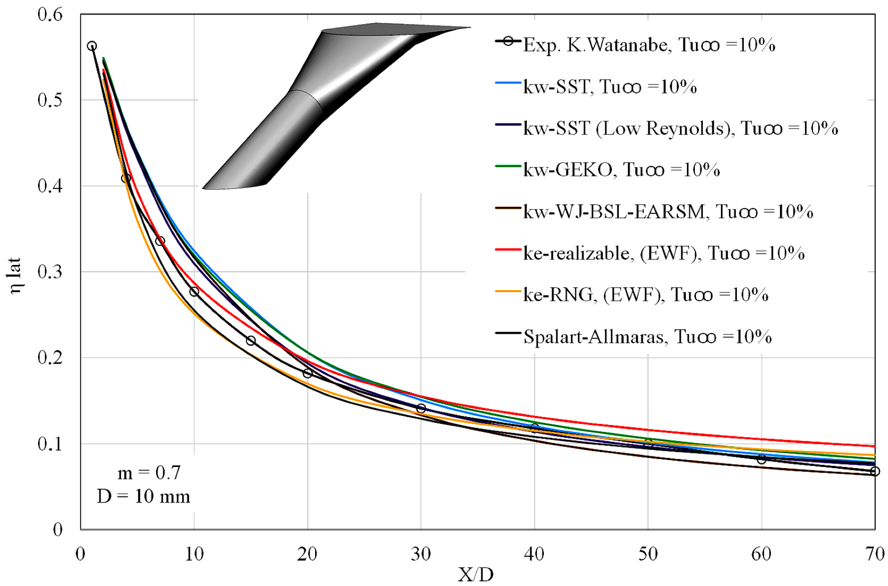
As can be seen from Fig. 8, *b*, the results for all turbulence models are significantly higher than the experimental data. The closest results to the experiment were shown by the models of the k-ε group (Realizable and RNG), although at large values of X/D, the calculation using the k-ω turbulence model WJ-BSL-EARSM has a significantly better result. Unfortunately, the popular k-ω – SST model demonstrates the worst agreement with the experiments. Additionally, a CFD calculation [19] from Iowa State University is presented, with which the validation of the base model of a cylindrical hole was carried out in work [20]. As can be seen from Fig. 8, *b*, this CFD calculation, which was made using the k-ε Realizable turbulence model, coincides well with the calculation carried out in this work with the same turbulence model.

Fig. 9, *a*, *b* shows the results of the calculation of the lateral efficiency of model 2, which is a fan-shaped hole, for turbulence intensity of 0.5% and 10%, respectively. The authors of paper [16] noted that for this type of hole the cooling efficiency depends weakly on the blowing ratio m and still remains high at an blowing ratio greater than 1.5. For the blowing ratio $m = 0.7$, the authors presented an analysis of the influence of the main flow turbulence.

All turbulence models for model 2 showed a satisfactory agreement with the experiment in both magnitude and trend, however, the k-ε Realizable turbulence model gives the closest absolute results of the cooling efficiency to the experimental data for both turbulence intensity of 0.5% and 10%. According to the experiment for model 2, at a higher turbulence intensity along the entire length X/D, a drop (on average by 0.04) in the cooling efficiency is observed. This relative drop in cooling efficiency with increasing turbulence is found to be better modeled by the k-ω models. The k-ε models (Realizable and RNG) are less sensitive to the drop in efficiency with increasing turbulence, while the one-parameter Spalart-Allmaras model exhibits the largest drop in efficiency when going from 0.5% to 10% turbulence.



a



b

Fig. 9. Calculations of film cooling efficiency on a level 3 grid:
a — comparison of calculations of lateral efficiency of model 2 (fan hole) at $m = 0.7$ with turbulence intensity of 0.5%;
b — comparison of calculations of lateral efficiency of model 2 (fan hole) at $m = 0.7$ with turbulence intensity of 10%

Fig. 10, *a* shows the results of the calculation of the lateral efficiency of model 3, which is a quasi-two-dimensional slot or oval hole. According to [16], the film cooling efficiency of this hole reaches a maximum at $m = 1.0$. Comparison of the experimental data with the experimental Weidhardt equation (3) for the slot showed good agreement with the experiment at large values of X/D; however, the proposed Hartnett correlation (4) generally demonstrates better agreement with the experiment:

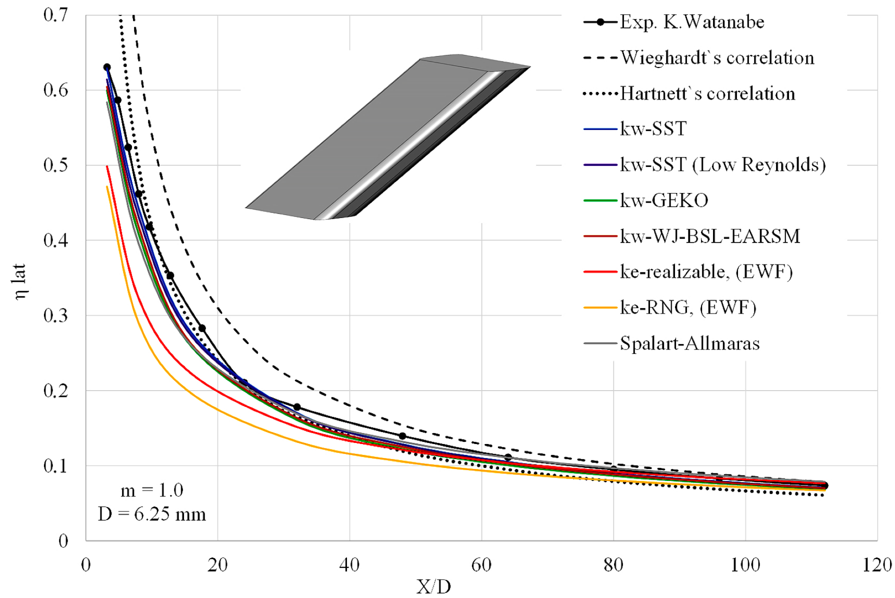
$$\eta_f = 21.8\left(X/MS\right)^{-0.8}, \tag{3}$$

$$\eta_f = 16.9\left(X/MS\right)^{-0.8}. \tag{4}$$

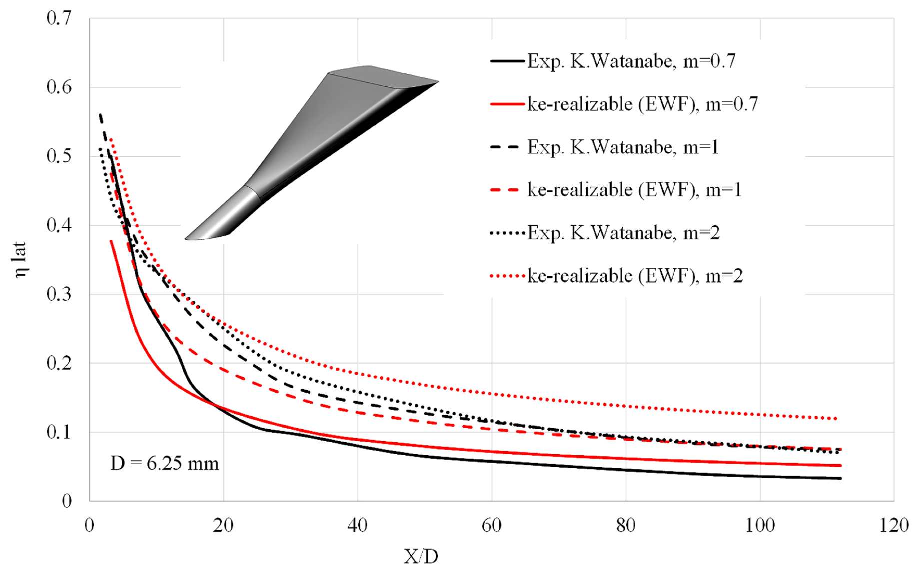
In this case, the width of the equivalent gap S was calculated by the authors of [16] as the ratio of the area of a conditional cylindrical hole, the diameter of which is 6.25 mm, to the pitch between the holes. Thus, the blowing ratio $m = 1$ is obtained by the coolant flow rate for this conditional hole.

All turbulence models of the k-ω group, as well as the Spalart-Allmaras model, agree well with the experimental data. The k-ε group models at small values of X/D showed clearly underestimated data.

Fig. 10, *b* shows the result of the calculation of the lateral efficiency of model 4, which is a diffuse slot. The diameter of the hole, equal to 6.25 mm in the initial section, expands laterally at angles of 15° and forms the initial oval part of the hole, as in model 3. The best results were provided by the k-ε Realizable turbulence model. The k-ω group models in this case demonstrate a rather noticeable asymmetric distribution of the cooling efficiency relative to the centerline, which in the stationary RANS problem can only be explained by the uneven distribution of the grid. The results for the k-ε Realizable turbulence model are given for the blowing ratios $m = 0.7$, $m = 1.0$ and $m = 2.0$.



a



b

Fig. 10. Calculations of film cooling efficiency on a level 3 grid: *a* – comparison of calculations of lateral efficiency of model 3 (oval hole) at $m = 1.0$; *b* – comparison of calculations of lateral efficiency of model 4 (diffuse slot) at $m = 0.7$, $m = 1.0$ and $m = 2.0$

In [16], experimentally obtained air temperature contours in the *YZ* plane at several X/D values were also presented, which were obtained by thermocouples installed on a moving comb. It was noted that in model 3, which is an oval hole, the maximum efficiency of the film is strictly behind the hole. A sharp drop in efficiency was also observed in the lateral direction immediately behind the edge of the hole of model 3 due to the mixing of the jet with the main flow. However, in model 4, which has the same geometry of the outlet section as model 3, the maximum efficiency is shifted to the side (to the left by the authors), which can be explained by the Coanda effect, that is, the jet breaking due to its sticking to the wall. A comparison of the film cooling efficiency of models 3 and 4 at the blowing ratio $m = 1$ (but with the same coolant flow rate) for the k-ε Realizable turbulence model is shown in Fig. 11.

The efficiency distribution pattern on a flat plate confirms the above observations by the authors of [16].

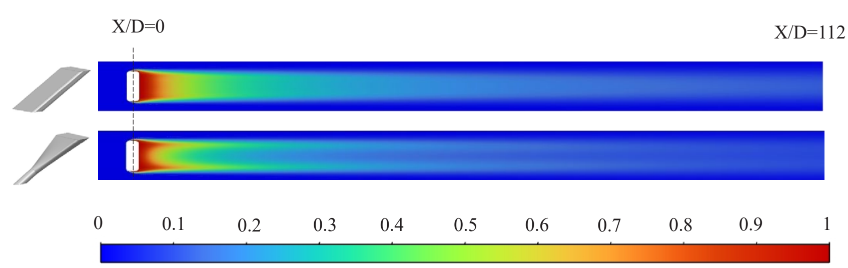


Fig. 11. Comparison of film cooling efficiency of models 3 and 4 with the same outlet cross section at the blowing ratio $m = 1$ with the turbulence model k-ε Realizable

To assess the operating range and stability of the working model, an additional calculation with boundary conditions [16] ($DR = 0.9$) was performed with the k-ε Realizable turbulence model for a 7-7-7 type orifice. The results obtained were compared with the experiment at $DR = 1.5$, with which the model was validated in [15]. This approach allows us to assess the performance of the model in the case of mismatch of individual physical parameters, but comparable injection modes. In [15], the authors also used the k-ε Realizable turbulence model to validate the calculation model. The 7-7-7 hole was located at an angle of $\alpha = 30°$, the length of the cylindrical part was 2.5D, and the length of the expanding section was 3.5D. The area of the output cross-section of the hole was 2.5 times larger than the area of the cylindrical channel. For comparison with the experiment under the same conditions, after the results obtained, the efficiency value for the model with $P / D = 5$ was reduced in proportion to the step. The comparison results are shown in Fig. 12.

The CFD calculation showed reasonable agreement with the experimental data, and, at the same time, it describes the trend well. The maximum deviation from the experimental data occurred for the curve $m = 0.5$. The average deviation along the X / D coordinates for this curve is 16%. The smallest deviation from the experimental data was shown by the curve for $m = 1.0$, the average deviation along the X / D coordinates of which is 7%.

5. 4. Analyzing the ability of the validated model to study the technological features of applying TBC

High-pressure gas turbine blades are often covered with a layer of ceramics that acts as a thermal barrier coating. The coating is applied to the blade after making perforation holes, including getting into the hole itself. It is obvious that such a technological sequence can lead to distortion of the outgoing air jet, changing the jet slope and affecting the efficiency of film cooling. That is why it can be proposed to make a trench in the TBC layer at the location of a number of holes. Making a trench with a small relative height H / D, which is far from the optimal values of the trench height in this case, prevents partial blocking of the hole and, as a result, jet deflection. These technological features of the production of film-cooled blades are the subject of further research, which would be impossible without careful validation of computational CFD models.

In this regard, simplified calculation models were additionally built for a possible option of blocking the TBC hole (Fig. 13, *a*) and a variant with a trench made in the TBC (Fig. 13, *b*). The 3-level grid was taken as a basis (Table 1).

A preliminary analysis of film cooling on a plate has revealed the influence of the TBC configurations on the film efficiency for a shaped hole. Figure 14 shows a comparison of the efficiency of the shaped hole 7-7-7, the variant with the TBC application after the hole is made, and the variant in which a trench is made in the TBC layer.

As can be seen from Fig. 14, at small blowing ratios, blocking the shaped hole does not harm the cooling efficiency. Making a trench in the TBC layer in this case is unnecessary and even worsens the cooling efficiency. At large blowing ratios, jet deflection due to blocking the TBC hole leads to a rather noticeable deterioration in the cooling efficiency. Making a trench in this case is justified.

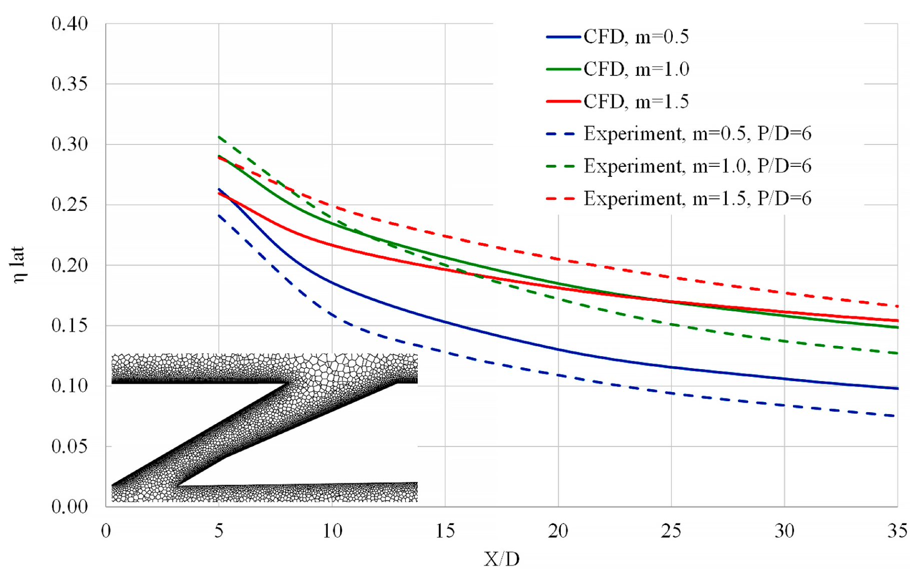
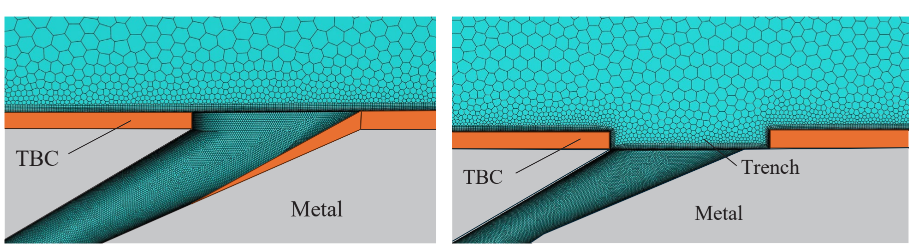


Fig. 12. Comparison of CFD calculation results using the k-ε Realizable turbulence model with experimental data



a *b*

Fig. 13. The selected computational grid of level 3 with modeling of the thermal barrier coating layer (together with the sublayer) $H / D = 0.4$: *a* – hole shape 7-7-7 with blocking by the thermal barrier coating at the outlet; *b* – hole shape 7-7-7 with a trench made inside the thermal barrier coating

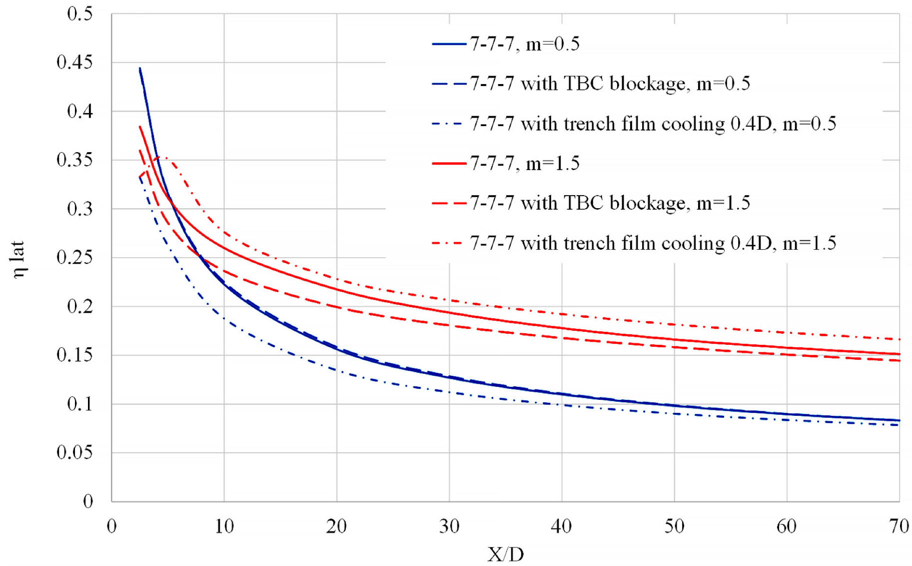


Fig. 14. Comparison of cooling efficiency results for a shaped hole with $P/D=5$ when applying TBC after hole execution and with the proposed trench

It is clear that even these initial results indicate the need to make adjustments to existing methods for predicting film performance.

6. Discussion of results based on the validation of the calculation model

To substantiate the selected model and to confirm reasonable accuracy in determining the efficiency of film cooling from cylindrical and shaped holes, a 3D CFD model on a flat plate was developed and validated. In the future, this will make it possible to optimize the geometry of the holes and injection modes in the cooling problems of gas turbine engine blades and provide recommendations for improving the film cooling system. The size of this model is sufficient for modeling a thin layer of thermal barrier coating both for describing partial blocking of the hole and for describing a trench made in the coating.

For the *k-ε* Realizable turbulence model, the best convergence of the results for lateral cooling efficiency was obtained already on a coarse grid of level 1 (Fig. 5, *a*). The same conclusion regarding this turbulence model was drawn in [9] when studying the mesh convergence for film cooling with injection into a trench. However, to calculate the film cooling efficiency for given geometric and operating parameters without loss of accuracy, only grids of 3 or 4 levels (Table 1) can be selected in the future, depending on the required geometry detail. Grids of 1 and 2 levels cannot be selected due to their significant impact on the film cooling efficiency along the center line (Fig. 5, *b*).

The structured block grid has no advantage over the polyhedral grid in modeling the lateral efficiency (Fig. 7, *a*). However, it showed a smaller excess of the film cooling efficiency values along the centerline (Fig. 7, *b*), namely by an amount of up to 0.03 over the entire length of X/D for a round hole at $m=0.5$.

The *k-ω* class models showed a better match of the lateral cooling efficiency for a cylindrical hole at small $X/D\leq 10$, especially at small blowing ratios (Fig. 8, *a*). However, the models of this class also demonstrated a greater excess of efficiency along the centerline over the experiment than the *k-ε* class models (Fig. 8, *b*). Thus, the popular *k-ω* SST model, with a good match with the experiment in lateral efficiency, showed the largest excess along the centerline (more than 0.2 over the entire range of X/D) among all models. It should be noted that the *k-ω* SST turbulence model with Low Reynolds corrections option showed better results than the basic *k-ω* SST.

In contrast to [8, 9], our study uses existing, unadapted turbulence models to calculate the efficiency of film cooling on a flat plate. This allows for a wider application of the CFD model, making it more versatile, but with a loss of accuracy in some criteria where an adapted model may be more useful.

None of the seven RANS turbulence models showed agreement on all criteria of the four orifice models. However, the most universal among the considered models is *k-ε* Realizable with enhanced wall function and standard constants C2-Epsilon = 1.9, TKE Prandtl number = 1, TDR Prandtl number = 1.2, Energy Prandtl number = 0.85, Wall Prandtl number = 0.85.

The *k-ε* Realizable turbulence model among all turbulence models provided the best agreement with the results of lateral efficiency of fan orifices, namely models 2 and 4 (Fig. 9, 10). This can be explained by the better distribution of turbulent stresses in the complex flow zones. Also, together with the *k-ε* RNG and *k-ω* WJ-BSL-EARSM turbulence models, the *k-ε* Realizable model showed the smallest error in determining the efficiency along the centerline of model 1. At the same time, the error in determining the lateral efficiency of model 1 at $m=0.5$ and $X/D>15$ for *k-ε* Realizable does not exceed 10%. Additionally, a calculation with boundary conditions [16] was performed with the *k-ε* Realizable turbulence model for a 7-7-7 type hole. The calculation showed a satisfactory agreement with the experimental data (Fig. 12).

The *k-ε* Realizable turbulence model cannot be recommended only for calculating the efficiency of the slot (model 3), although it describes the trend quite well (Fig. 10, *a*).

The above allows us to state that for these geometric and operating parameters the most universal CFD model of film cooling was developed, which is justified for further work.

In further theoretical studies using this CFD model, the following limitations must be taken into account, within which this validation was carried out:

– the model is valid for cylindrical and fan-shaped holes with expansion angles up to 15° in the lateral and front directions;

– injection parameters m = 0.5–2.0;

– ratio of coolant density to main flow density DR = 0.9 – 1.5;

– main flow turbulence intensity Tu_{∞} = 0.5–10%;

– longitudinal relative length X / D = 2–112;

The disadvantages of the study include:

– underestimation of the lateral efficiency of the cylindrical hole relative to the experiment, especially at high injection parameters $m \geq 1.5$;

– conducting research at low temperatures of the coolant and the main flow, which may require adjustment by the ACR parameter (advective (heat) capacity ratio [21]) when scaling to real engine parameters;

The development of this research may consist of the following:

– validation has been performed for a flat plate, but real blade shapes (suction side and pressure side) require additional calculations for convex and concave surfaces;

– performing calculations with different angles β, including with backward holes directed towards the flow.

7. Conclusions

1. Mesh convergence study showed that the results of lateral efficiency and centerline cooling efficiency for given geometric and operating parameters can be sufficiently reliable only when using a level 3 mesh with a size of 5.8 million elements or higher.

2. The comparison of the unstructured computational mesh with the structured block mesh demonstrates the absence of a difference in calculating lateral efficiency. The advantage of the structured mesh in calculating centerline efficiency is not significant. For further studies, a simpler polyhedral mesh was chosen.

3. None of the considered turbulence models showed a coincidence in all criteria of the four hole models. However, the most universal among the seven turbulence models is the k-ε Realizable with an enhanced wall function, which is recommended for further studies for cylindrical and shaped holes.

4. The thorough validation of CFD mesh models allowed us to lay a reliable basis for studying the influence of technological features of applying TBC in the locations of perforation holes of gas turbine blades. The difference in the efficiency of film cooling of a shaped hole with the usual technology of applying TBC and the variant with a trench in TBC can reach 0.05. At modern gas temperature levels, this value can be up to 50°C when calculating the film temperature.

Conflicts of interest

The authors declare that they have no conflicts of interest in relation to the current study, including financial, personal, authorship, or any other, that could affect the study and the results reported in this paper.

Funding

The study was conducted without financial support.

Data availability

All data are available in the main text of the manuscript.

Use of artificial intelligence

The authors confirm that they did not use artificial intelligence technologies when creating the current work.

Authors' contributions

Oleg Shevchuk: Methodology, Formal analysis, Investigation, Visualization, Writing – original draft, Writing – review & editing; **Oleksandr Tarasov**: Conceptualization, Validation, Supervision.

References

1. Jauregui, R., Silv, F. (2011). Numerical Validation Methods. Numerical Analysis – Theory and Application. https://doi.org/10.5772/23304
2. Shevchuk, O., Tarasov, O. (2025). Review of the Current State of Problem of Film Cooling for Gas Turbine Blades. NTU "KhPI" Bulletin: Power and Heat Engineering Processes and Equipment, 1, 29–44. https://doi.org/10.20998/2078-774x.2025.01.03
3. Bunker, R. S. (2010). Film Cooling: Breaking the Limits of Diffusion Shaped Holes. Heat Transfer Research, 41 (6), 627–650. https://doi.org/10.1615/heattransres.v41.i6.40
4. Shevchuk, O. V. (2025). Analysis of Modern Numerical Approaches to Film Cooling Simulation on a Flat Surface: Trends, Errors and Correlation Dependencies. Journal of Mechanical Engineering, 28 (4), 11–25. https://doi.org/10.15407/pmach2025.04.011
5. Chen, X., Li, J., Long, Y., Wang, Y., Weng, S., Yavuzkurt, S. (2020). A Conjugate Heat Transfer and Thermal Stress Analysis of Film-Cooled Superalloy With Thermal Barrier Coating. Volume 7B: Heat Transfer. https://doi.org/10.1115/gt2020-16241
6. Zamiri, A., You, S. J., Chung, J. T. (2020). Numerical Evaluation of Surface Roughness Effects on Film-Cooling Performance in a Laid-back Fan-Shaped Hole. Volume 7B: Heat Transfer. https://doi.org/10.1115/gt2020-14525
7. Barigozzi, G., Zamiri, A., Brumana, G., Franchina, N., Chung, J. T. (2025). On the impact of Reynolds number on the performance of a trenched shaped hole. 16th European Conference on Turbomachinery Fluid Dynamics and Thermodynamics. https://doi.org/10.29008/etc2025-107
8. Petelchic, V. Yu., Halatov, A. A., Pismenniy, D. N., Dashevskiy, Yu. Ya. (2013). Adaptation of SST turbulence model for a flat plate film cooling simulation. Eastern-European Journal of Enterprise Technologies, 3 (12 (63)), 25–29. Available at: https://journals.uran.ua/eejet/article/view/14874/
9. Fischer, L., James, D., Jeyaseelan, S., Pfitzner, M. (2023). Optimizing the trench shaped film cooling design. International Journal of Heat and Mass Transfer, 214, 124399. https://doi.org/10.1016/j.ijheatmasstransfer.2023.124399

10. Liu, C.-L., Zhu, H.-R., Bai, J.-T. (2008). Effect of turbulent Prandtl number on the computation of film-cooling effectiveness. International Journal of Heat and Mass Transfer, 51 (25-26), 6208–6218. https://doi.org/10.1016/j.ijheatmasstransfer.2008.04.039
11. Shih, T.-H., Liou, W. W., Shabbir, A., Yang, Z., Zhu, J. (1995). A new k-ε eddy viscosity model for high reynolds number turbulent flows. Computers & Fluids, 24(3), 227–238. https://doi.org/10.1016/0045-7930(94)00032-t
12. Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal, 32 (8), 1598–1605. https://doi.org/10.2514/3.12149
13. Diskin, B., Thomas, J. (2012). Effects of mesh regularity on accuracy of finite-volume schemes. 50th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition. https://doi.org/10.2514/6.2012-609
14. Furgeson, M. T., Flachs, E. M., Bogard, D. G. (2025). Adjoint Optimization of Gas Turbine Film Cooling Geometry With Elevated Mainstream Mach Number. Volume 5: Energy Storage; Fans and Blowers; Heat Transfer: Combustors; Heat Transfer: Film Cooling. https://doi.org/10.1115/gt2025-154287
15. Avcun, S., Erdem, E., Sal, S., Yasa, T. (2025). The Effect of Crossflow Velocity on the Film Cooling Effectiveness of Fan Shaped Holes. Volume 5: Energy Storage; Fans and Blowers; Heat Transfer: Combustors; Heat Transfer: Film Cooling. https://doi.org/10.1115/gt2025-154073
16. Watanabe, K., Matsuura, M., Suenaga, K., Takeishi, K. I. (1999). An experimental study on the film cooling effectiveness with expanded hole geometry. Proceedings of the international gas turbine congress. Kobe, 615–622.
17. Celik, I., Ghia, U., Roache, P., Freitas, C., Coleman, H., Raad, P. (2008) Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. Journal of Fluids Engineering, 130 (7), 078001. https://doi.org/10.1115/1.2960953
18. Pedersen, D. R., Eckert, E. R. G., Goldstein, R. J. (1977). Film Cooling With Large Density Differences Between the Mainstream and the Secondary Fluid Measured by the Heat-Mass Transfer Analogy. Journal of Heat Transfer, 99 (4), 620–627. https://doi.org/10.1115/1.3450752
19. Na, S., Shih, T. I.-P. (2006). Increasing Adiabatic Film-Cooling Effectiveness by Using an Upstream Ramp. Journal of Heat Transfer, 129 (4), 464–471. https://doi.org/10.1115/1.2709965
20. Hussain, S., Yan, X. (2020). Implementation of Hole-Pair in Ramp to Improve Film Cooling Effectiveness on a Plain Surface. Volume 7B: Heat Transfer. https://doi.org/10.1115/gt2020-14838
21. Rutledge, J. L., Polanka, M. D. (2014). Computational Fluid Dynamics Evaluations of Unconventional Film Cooling Scaling Parameters on a Simulated Turbine Blade Leading Edge. Journal of Turbomachinery, 136 (10). https://doi.org/10.1115/1.4028001