Simulation and Experimental Analysis of the Temperature Field of Flow of Airport Snow Thrower Based on CFD Theory

Zhenglei Chen*, Wantong Cai, Liangcai Cai, Guoping Cen

Department of Airport Construction Engineering, Air Force Engineering University, Xian, Shaanxi province, 710038, China

*Corresponding author’s e-mail: czl1126988739@163.com

Abstract: It is a hot issue for airport users and researchers to pay attention to the influence of the hot air flow generated by the hot blowing snow thrower on the durability and service life of the pavement. In order to clarify the influence of hot air flow generated by hot blowing snow throwers on the temperature distribution of airport runway, this paper studies the temperature distribution of airport runway under the action of hot air flow generated by hot blowing snow throwers by CFD simulation and design of field measurement experiments. The results show that the maximum temperature of the surface obtained by the simulation method is 554.21K, the limit temperature of the surface is 539.75K and the average temperature difference is 22.64K, which verifies the accuracy of the simulation model. In view of the research results, in order to promote the application of this type of snow thrower in the airport, it is necessary to verify whether the airport roadway can withstand the action of high temperature airflow of 539.75K.

1. Introduction
The airport runway is an important component of the airport flight zone. The strength and durability of the airport runway are important indexes to evaluate the airport operation status, which directly affect the service life of the airport. Compared with highway facilities, the airport runway not only bears more instantaneous load, but also is affected by the impact of aircraft landing and the high-temperature high-speed airflow from the engine during takeoff and landing.

Long time impact and high temperature will seriously reduce the pavement durability, thus shortening the service life of airport facilities. Therefore, it is of great significance to understand the influence of high temperature airflow on the pavement for the selection of pavement materials, durability assessment and maintenance.

At present, scholars at home and abroad mainly adopt two methods to study the temperature field distribution of the passage surface under the action of high temperature airflow: field measurement and simulation calculation. In terms of field measurement experiments, J. Godoy [1] using a low-cost wireless temperature sensor design the pavement temperature monitoring system, to monitor the temperature of pavement and the test pavement to evaluate system, verify the feasibility of using the pavement temperature sensor, but wireless sensor signal weakened gradually with the increase of the depth of pavement, lead to the error increases with the increase of the depth of pavement. B.-B. Teltayev [2] measured the road surface temperature and humidity performance based on the sensor, and analyzed the change characteristics of the road surface temperature with time and the distribution of temperature in depth at different moments. S. Cai [3] designed an intelligent sensor network structure, which is used to measure the temperature of the pavement structure. The effectiveness and reliability of the system are verified by experiments.
With the development of the computer industry, computational fluid dynamics (CFD) [4]-[5], as a simulation method for studying tail-air flow, has developed rapidly, and the method of using computer simulation to calculate the temperature field distribution under the action of tail-air flow has been realized. H. Wang et al. [6] used CFD method to conduct experimental research on the exhaust flow field of jet engine (hereinafter referred to as jet), and the results showed that the pressure and temperature of jet center gradually decreased with the increase of the distance between jet and nozzle. L.A.Benderskiy [7]-[8] et al. used the RANS/ILES method to simulate the tail jet flow of a double-cone supersonic aircraft, to detect the influence of co-flow velocity and distance on the temperature, pressure fluctuation and axial velocity of the jet danger area, and to study the influence of natural wind speed on the jet danger area. Yue Kuizhi [9] - [10] et al. numerically simulated the jet impact on flight deck and deflector of carrier-borne aircraft during takeoff by using standard k - equation and three-dimensional n-s equation based on CFD theory, and obtained the temperature characteristic distribution on the axis. Wei Zhiqiang et al. [11] summarized and analyzed the research status of simulation calculation methods of tail jet flow, summarized the characteristics and main problems of typical calculation models, and analyzed the precision and application of different models. Song Ma et al. [12] established a simulation model of jet flow at the tail of a carrier-mounted aircraft, studied the influence of high temperature, high speed and high pressure jets at different deflection angles on the guide plate, and compared the simulation results with the experimental results. Based on CFD calculation method, Fu - Dong Gao [13] et al. used four different turbulence models, namely shear stress transport (SST) k-w, standard k-w, standard k - and Reynolds stress model (RSM), to compare and verify the correctness of the numerical calculation method for gas jet flow of a single aircraft engine. The above research shows that it is feasible to study the temperature field distribution under the action of tail air injection flow by numerical simulation. Considering the need of measurement accuracy, the numerical simulation can be compared with the field measurement results to simplify the measurement process and verify the accuracy of the measurement results.

In recent years, with the continuous development of airport facilities, snow throwers, as an important snow thrower, have been widely used in many airports. Among them, the jet road surface sweeper is more widely used. This kind of snowblower can better achieve the purpose of snow removal and ice removal through high temperature high-speed airflow. However, there are few studies on the influence of hot air produced by jet road sweeper on the road surface. In this paper, based on CFD theory, the influence of hot air flow generated by jet road sweeper on the temperature field distribution of cement concrete pavement was numerically simulated, and field test experiments were designed and carried out. The experimental results were compared with the numerical simulation results to further verify the accuracy of the numerical simulation method. It is hoped that this can provide a basis for the application of jet road sweeper in the field of airport, and also facilitate the airport management to carry out precise maintenance and repair of road surface.

2. Model establishment and solution

2.1 Theoretical basis

With the rapid development of computer computation, the numerical algorithm which USES computer to solve the physical problem accurately has been developed rapidly. In the field of fluid mechanics, the combination of theoretical fluid mechanics and experimental fluid mechanics leads to the development of computational fluid dynamics (CFD), which provides a new way of thinking and means for us to study the influence of various engine tail jet flows on airport runway. CFD can be used to express the characteristics of the flow field of the nozzle in the form of mathematical equation. Using the three-dimensional rectangular coordinate system, the conserved n-s (navier-stokes) control equations can be expressed as:

Fluid continuity equation:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{W}) = 0 \tag{1}
\]
Fluid motion equation:

$$\frac{\partial \rho \bar{W}}{\partial t} + \nabla \cdot (\rho \bar{W} \otimes \bar{W} + p[I] - \bar{V} \cdot [\bar{\tau}] + \rho \{2\bar{\omega} \times \bar{W} + \bar{\omega} \times (\bar{\omega} \times \bar{r})\} = 0 \quad (2)$$

Fluid energy equation:

$$\frac{\partial \rho E}{\partial t} + \nabla \cdot (\rho E\bar{W} + p[I] \bar{W}) - \nabla \cdot (\kappa \nabla T + [\bar{\tau}] \bar{W}) - \rho \nabla \left(\frac{\bar{\omega}^2 \bar{r}^2}{2}\right) \cdot \bar{W} = 0 \quad (3)$$

Where, $\rho$ is the density of the fluid; $\bar{W}$ is the relative velocity vector of the fluid; $p$ is the static pressure of the fluid; $[I]$ is the unit tensor of the fluid; $[\bar{\tau}]$ is the viscous stress tensor of the fluid; $E$ is the total energy of the fluid; $\bar{\omega}$ is the rotation angular velocity of the fluid; $\bar{r}$ is the corresponding vector radius; $\kappa$ is the thermal conductivity of the fluid; $T$ is the temperature of the fluid.

Among them:

$$E = e + \frac{|W|^2}{2} \quad (4)$$

Where, $e$ is the internal energy of the fluid.

When equations (1), (2) and (3) are homogenized, the n-s governing equation is changed to the Reynolds average n-s system, which contains an additional pulse value. In this equation, the fluctuation value is an unknown quantity, so it is necessary to establish turbulence models related to various fluctuation quantities with the help of experience, so as to close the system of equations, so as to obtain analytical solutions.

The classical turbulence model includes zero equation, one equation and two equation. Because the zero equation and one equation model are too simple, the calculation of turbulent fluid is not accurate enough. So, at present, the fluid calculation mostly uses the two equation model. Among them, the $\kappa$-$\varepsilon$, $\kappa$-$\omega$, SST and SAS-SST turbulence models are most widely used. In this paper, the two-equation model of SST which is widely used in engineering problem calculation is adopted.

Because $\kappa$-$\omega$ turbulence model is relatively sensitive to the change of free fluid initial parameters of the shortcomings, SST 2 equation model will be $\kappa$-$\omega$ turbulent model and $\kappa$-$\varepsilon$ turbulence model, by mixing function F1 combination, in the near wall flow field using $\kappa$-$\omega$ turbulence model, in the far away from the wall using $\kappa$-$\varepsilon$ turbulence model, effectively play the advantages of the two models. The specific description is as follows:

$$\frac{\partial (\rho \bar{\omega})}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{\omega})}{\partial x_j} = P_\omega \left( \frac{M_e}{Re} \right) - \beta \rho \omega^2 \frac{Re}{M_e}$$

$$\frac{\partial (\rho \bar{\varepsilon})}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{\varepsilon})}{\partial x_j} = P_e \frac{M_e}{Re} - \beta \rho \varepsilon$$

The expression of eddy viscosity is

$$\mu_r = \min \left[ \frac{\rho}{\varepsilon} \bar{\mu} \left( \frac{Re}{M_e} \right) \right] \quad (7)$$

Among them:

$$S = \sqrt{2S_x S_y}, \quad S_x = 1 \left( \frac{\partial \mu_x}{\partial x} + \frac{\partial \mu_y}{\partial y} \right)$$

The Reynolds stress is expressed as:

$$\tau_y = 2\mu_r (S_y - \frac{\partial \mu_y}{\partial x} - \frac{\partial \mu_y}{\partial x}) - \frac{2}{3} \rho \bar{\omega} \delta_y \quad (8)$$
The approximate expression of the generation term of turbulent kinetic energy and specific dissipation rate is:

\[ P_e = r_x \frac{\partial u_i}{\partial x_j} \mu_s S^2 \quad (9) \]

\[ P_w = \frac{\gamma P_e}{\mu_f} \quad (10) \]

2.2 The geometric model

This section mainly studies the influence of jet from the nozzle of jet snowblower on the temperature field of the runway. Therefore, the outlet of jet snowblower and a space including the runway in the rear can be selected for the establishment of the calculation model.

Under the working condition of snowblower, there is an included Angle between the engine axis and the airport runway surface, so the nozzle outlet airflow will form an included Angle with the airport runway surface. By actual measurement, the included Angle between the engine axis and the runway surface is about 10° (as shown in figure 1).

The calculation space region adopts the three-dimensional cartesian coordinate system, the coordinate origin (0,0,0) is located at the center of the nozzle section. In the actual numerical simulation, the calculation space area of the model includes the outlet of the snowblower and the space area affected by the airflow after the outlet. In order to satisfy the far-field conditions, an area with a length of 30m and an outlet height of 5m is taken as the computational space area of the entire flow field in this paper (as shown in FIG. 2). The Angle between the mesh inlet section and the channel surface is 80°, which satisfies the condition that the engine axis and the channel surface are at a 10° Angle.

2.3 Grid division

The dividing method and mesh quality of network model have great influence on the numerical simulation results. It is important to reduce the number of meshes and improve the precision and efficiency of computation to adopt the structural meshing method as far as possible. In this paper, the whole computing region is divided into several sub-regions by block grid method, and the grids are generated respectively. The meshes in the inner, outlet and near wall area of the nozzle are locally...
2.4 Solution of model
The inlet boundary conditions of the space region are calculated according to the rated state of the engine, with a uniform total temperature of 673.15K and a speed of 530m/s. The outlet boundary conditions are open boundary conditions. The distribution of the outlet static pressure is determined by the radial equilibrium equation. The solid wall adopts adiabatic roughness boundary condition.

In order to solve the three dimensional Navier-Stokes equations of conservation form in relative coordinate system, the finite volume method based on unit center is used for difference method, and the implicit propulsion algorithm is used for time advance method. SST model is adopted for the turbulence model, which USES the standard model κ-ω in the low Reynolds number region inside the boundary layer and the model κ-ε in the high Reynolds number region outside the boundary layer, so that the equation is suitable for both the near and far wall surfaces. In the governing equation, the convective term is discretized by the second-order upwind scheme, and the time integral is carried out by the second-order backward euler equation. The turbulence equation is solved by the first-order upwind scheme and the turbulence variable is solved by the finite-order implicit euler equation.

In this paper, the convergence conditions are as follows:
(1) the residual of the total flow decreases to less than 10;
(2) the relative value of flow residuals of nozzle and inlet and the total flow residuals of the whole site is less than 1%;
(3) as the number of iteration steps increases, the total performance parameters no longer change significantly, and the parameters at a certain point in the flow field remain unchanged.

2.5 Model validation

2.5.1 Grid independence test
The time step was set to 0.001s, and numerical calculation was carried out for three calculation grids with the number of grids of 3 million, 5 million and 7 million, respectively. When the pavement temperature reached the steady state, the limit temperature of the pavement was recorded. The results are shown in table 1.

| The grid number | 3000000 | 5000000 | 7000000 |
|-----------------|---------|---------|---------|
| Maximum temperature(K) | 552.98 | 554.21 | 554.54 |

As can be seen from the table, when the number of grids exceeds 5 million, the maximum temperature difference of the channel is not big. In the end, the total number of computational space region grids selected in this paper is 5 million, which can guarantee the computational efficiency on the premise of satisfying the computational accuracy.

2.5.2 Time independence test
The number of grids was set to 5 million, and the time step was 0.005s, 0.001s and 0.0001s, respectively. The time independence test was carried out on the number of selected grids. After steady state simulation, the maximum temperature obtained was shown in table 2.

| Time step(s) | 0.005 | 0.001 | 0.0001 |
|--------------|-------|-------|--------|
| Maximum temperature(K) | 554.96 | 554.21 | 553.79 |

As can be seen from the table, when the number of grids is constant, the time step has no obvious influence on the maximum temperature. After comprehensive consideration, the time step is 0.001s.

2.6 Results analysis
When the jet flow of snowmobile impinges on the airport runway, the temperature of the runway will
increase significantly, so as to achieve the purpose of snowblowing and snowmelt. FIG. 4 shows the temperature cloud diagram of the x-y plane perpendicular to the channel plane at different distances.

As the airflow jetted backward at a certain Angle, the jet stream soon approached the runway, making the temperature of the runway rise. When \( Z = 0.75 \) m, is the tail jet flow at the nozzle outlet, which has not yet reached the airport runway surface, and the airflow temperature is high; When \( Z = 1.25 \) m, the tail jet stream jetted to the airport runway and conducted heat exchange with the runway to increase the temperature of the runway. When \( Z = 1.75 \) m, the core high temperature area in the hot tail jet stream jetted to the airport runway area at the maximum, so that the temperature difference between the airflow and the runway was the largest, the airflow temperature on the runway surface reached the peak, and the runway temperature rose the fastest. When \( Z = 2.25 \) m, due to heat dissipation, the temperature in the core high temperature area decreases, the heat transferred by the airflow to the channel surface decreases, and the airflow temperature on the channel surface decreases.

FIG. 4 shows the temperature cloud image of the spanwise cross section

FIG. 5 shows the cloud map of the temperature distribution on the airport roadway surface after the airflow generated by snowmobile is acted on by numerical calculation. According to the analysis in FIG. 4, the surface temperature of the track surface tends to rise first and then fall under the action of high temperature airflow. As can be seen from the graph in figure 6, the air flow temperature on the surface of the channel starts to rise from 0.7 m away from the outlet of the nozzle, and reaches the peak at the temperature near 2 m, and the peak temperature is 554.21 K. After that, the air flow temperature on the surface of the channel does not increase any more. As the distance from the nozzle increases, the airflow temperature on the surface of the channel gradually decreases, and finally approaches the ambient temperature.
3. Field measurement experiment

3.1 Test preparation
Before the experiment is used to simulate precast concrete pavement environment, sensors embedded in the middle of the test pavement concrete precast slab location, layout points altogether 13, interval 50 cm, 6, 10, sensors under the surface of 10 mm, 7 sensor under the surface of 20 mm, the rest of the 10 sensors buried under the pavement surface about 2 mm position, converter embedded in the corresponding concrete slab, test point distribution is shown in figure 7.

3.2 Field test
The snowblower is a wp5 engine, and the nozzle is 23cm above the runway. The operation state of snowblower is only rated state, which is selected for test. The snowblower is parked at the edge of the apron surface at 50cm. As the engine of the snowblower is installed at a low position, the position where the airflow touches the ground is close to the nozzle. The field test is shown in figure 8~ figure 9.

The test steps are as follows:

1. Dig out the turf soil within the range of 4.05m in length, 3.04m in width and 5cm in thickness at the edge of the apron, embed the plastic box in the position where the temperature converter is needed, and dig a wire trench.

2. Prefabricated panels with sensors are arranged on the ground, with an interval of about 1cm between them. Connect the wire and debug it. After it is correct, lay the other concrete prefabricated board and bury the wire under the board.

3. After debugging the instrument and there is no problem, fill the gap with cement mortar. In order to prevent the prefabricated plate from being blown up by air, concrete 50cm wide and 5cm thick should be poured on site on each side, and water should be sprayed for curing over 2d.
(4) When the snowblower is running, the engine is in rated state. After the test, the temperature values at each measuring point during the 2min operation of the engine in rated state are recorded. There are two tests in total.

Figure 8. Laying of concrete prefabricated slabs

Figure 9. Test status of jet snowblower

3.3 Field test results and analysis

3.3.1 The test results
The measurement point no. 6 was 1.25m away from the nozzle. The temperature curve of the measurement point over time in the test is shown in FIG. 10.

Figure 10. Temperature time curve of measuring point 6

The maximum temperature measured at the measuring point with the same embedding depth on the axis of the track surface is calculated, as shown in table 3. The maximum temperature measured at different depths at the same position on the trace surface is calculated, as shown in table 4.

| Measuring point | 2      | 5      | 11     | 13     |
|-----------------|--------|--------|--------|--------|
| Distance (m)    | 0.75   | 1.25   | 1.75   | 2.25   |
| Temperature (K) | 332.55 | 439.45 | 539.75 | 521.45 |

| Measuring point | 1.25 | 1.75 |
|-----------------|------|------|
| Distance (m)    | 5    | 6    | 7    | 11   | 10   |
| Depth (mm)      | 2    | 10   | 20   | 2    | 10   |
| Temperature (K) | 439.45 | 426.75 | 417.85 | 539.75 | 520.75 |

3.3.2 Test result analysis
(1) The nozzle of snowblower is very close to the road surface, and the airflow reaches the surface of the road very quickly, so that the temperature of the road surface increases, and the horizontal
distribution is high in the middle and low on both sides.

(2) It can be seen from the curve of the temperature change with time in figure 9 that the temperature value of the middle road surface at the measuring point no. 6 keeps increasing with the passage of time, and the road surface temperature rises rapidly and the temperature value is high. In about two minutes, the surface temperature tends to be stable and does not continue to increase. Therefore, the surface temperature measured at each measuring point in two minutes is taken as the maximum temperature at this position under the action of high temperature airflow.

(3) According to analysis table 3, under the action of high temperature airflow generated by snowblower, the temperature of the runway first increases and then decreases with the increase of distance in the axial direction. The maximum temperature of the runway reaches 539.75K around 1.75m from the nozzle.

(4) According to table 4, when the distance from the nozzle of the snowblower is the same, the temperature of the road surface gradually decreases with the increase of the embedding depth.

4. Results comparative analysis

The surface temperature values of the airfield runway from the engine nozzle Z=0.75m, 1.25m, 1.75m and 2.25m of numerical simulation were compared with the maximum temperature of the runway at the same location measured in the field measurement experiment. The statistical data are shown in table 5. Based on this, the temperature comparison diagram of the Central Line trace plane is drawn, as shown in FIG. 11.

| Distance (m) | 0.75  | 1.25  | 1.75  | 2.25  |
|-------------|-------|-------|-------|-------|
| Experimental temperature (K) | 332.55 | 439.45 | 539.75 | 521.45 |
| Simulation temperature (K)    | 376.63 | 456.32 | 554.21 | 536.59 |
| Temperature difference (K)    | 44.08  | 16.87  | 14.46  | 15.14  |

Figure 11. Center line trace surface temperature contrast diagram

By analyzing table 5 and figure 10, it can be concluded that:

(1) The temperature field distribution obtained by simulation experiment is roughly the same as that obtained by field experiment. Both of them show the rule of increasing first and then decreasing on the axis, and both of them reach the maximum temperature near the nozzle of 1.75m. Comparatively speaking, the maximum temperature measured in the experiment is closer to the engine nozzle than the maximum temperature obtained by simulation.

(2) The simulation value is larger than the test value on the whole. Within the scope of the diagram, the maximum temperature error is 44.08K when Z=0.75m, and the minimum temperature error is 14.46K when Z=1.75m, with an average error of 22.64K. Analysis reason: the simulation situation is relatively ideal, but there are environmental factors and measurement errors in the field test. Moreover, the simulation results correspond to the surface temperature of the airport runway, while in the experiment, the temperature at the depth of 2mm of the airport runway is collected by the sensor.
Therefore, it is normal that the simulation results are higher than the results obtained in the field test.

5. Conclusion
In this paper, the distribution of the temperature field on the airfield under the action of hot blowing snow thrower is studied by the method of simulation calculation and field measurement. By comparing the simulation results with the field measurement results, it can be found that under the action of high temperature airflow, the surface temperature increases first and then decreases along the direction of the airflow axis. The surface temperature measured by the two methods reaches the maximum near the airflow nozzle of the snow thrower at 1.75m. The maximum temperature calculated by the simulation experiment is 554.21K, the maximum temperature measured by the field measurement experiment is 539.75K, and the average temperature difference between the two methods is 22.64K, which verifies the accuracy of the simulation model. At the same time, it provides a basis for airport users to promote the application of this kind of snow thrower. Before the application, it is necessary to verify whether the airport runway can withstand the action of high temperature airflow of 539.75K.

Acknowledgments
This work was financially supported by The National Natural Science Foundation of China (ID: 51578540 and ID: 51608526).

References:
[1] J. Godoy, R. Haber, J. Muñoz, F. Matía, and Á. García. (2018) Smart Sensing of Pavement Temperature Based on Low-Cost Sensors and V2I Communications. Sensors., vol.18, no.7, 2092.
[2] B.-B. Teltayev, and E.-A. Suppes. (2019) Temperature in pavement and subgrade and its effect on moisture. Case Studies in Thermal Engineering., vol.13, 100363.
[3] S. Cai, H. Yuan, Y. Cui, B. Tian, and J. Lv. (2016) An ISO/IEC/IEEE21451 smart sensor network for distributed measurement of pavement structural temperature. International Journal of Distributed Sensor Networks., vol.22.
[4] J. D. Anderson. (1995) Computational Fluid Dynamics, New York, NY, USA: McGraw-Hill.
[5] D. Caughey, M. Hafez. (2005) Frontiers of Computational Fluid Dynamics. Singapore: World Scientific.
[6] H. F. Wang, L. C. Cai, and X. L. Chong, Hao Geng. (2015) Experimental Study of the Jet Engine Exhaust Field of Aircraft and Blast Fences. Promet–Traffic&Transportation., vol.27, no.2, pp.181-190.
[7] L. A. Benderskiy, D. A. Lyubimov, and A. O. Chestnykh. (2018) NUMERICAL INVESTIGATION ON THE INTERACTION OF A PAIR OF HOT OFF-DESIGN SUPERSONIC JETS WITH A JETBLAST DEFLECTOR. TsAGI Science Journal., vol. 49, no.1, pp.13–28.
[8] L.-A. Benderskii, D.-A. Lyubimov, A.-O. Chestnykh, B.-M. Shabanov, and A.-A. Rubakoval. (2018) The Use of the RANS/ILES Method to Study the Influence of Coflow Wind on the Flow in a Hot, Nonisobaric, Supersonic Airdrome Jet during Its Interaction with the Jet Blast Deflector. High Temperature., vol. 56, no.2, pp.247-254.
[9] K.-Z. Yue, L.-L. Cheng, H. Liu, and Y.-L. Wang. (2015) Analysis of jet blast impact of embarked aircraft on deck takeoff zone. Aerospace Science and Technology., vol.45, pp. 60-66
[10] K.-Z. Yue, Y.-C. Sun, W. Liu, and W.-G. Guo. (2015) Analysis of the Flow Field of Carrier-Based Aircraft Exhaust Jets Impact on the Flight Deck. International Journal of Aeronautical and Space Sciences., vol.16, no.1, pp.1-7.
[11] Z.-Q. Wei, Q.-L. Qu, W. Liu, X.-H. Xu. (2019) Review on the artificial calculating methods for aircraft wake vortex flow field parameters. ACTA AERODYNAMICA SINICA, Vol.37, No.1, p.33-41.
[12] M. Song, J.-G. Tan, X.-K. Li, and J. Hao. (2018) The effect analysis of an enginejet on an aircraft blast deflector. Transactions of the Institute of Measurement and Control., pp.1-12.
[13] F.-D.Gao, D.-X.Wang, H.-D.Wang, M.-M.Ji. (2018) Numerical Analysis and Verification of the Gas Jet from Aircraft Engines Impacting a Jet Blast Deflector. CHINESE JOURNAL OF MECHANICAL ENGINEERING. Vol.31, No.1, pp.1-11.