

Effects of Reynolds number and layout on aerodynamic and heat transfer characteristics of an aluminum sheet treated in a gas-cushion furnace

Licheng Wu^{1,2}, Can Kang^{1*}, Patrick G. Verdin², Jin Yin¹, Yu Xie³

1. School of Energy and Power Engineering, Jiangsu University, Zhenjiang 212013, China

2. Faculty of Engineering and Applied Sciences, Cranfield University, Cranfield MK43

0AL, United Kingdom

3. State Key Laboratory of Clean Energy Utilization, Zhejiang University, Hangzhou 310027, China

Abstract:

Heat treatment of an aluminum sheet through a gas-cushion furnace is numerically investigated. The sheet of 0.6 mm thick was suspended due to simultaneous impingement of upward and downward gas jets. Effects of Reynolds number ($Re = 8,000-32,000$) and upper/lower distance ratios (3:5, 1:1, 5:3) on the flow and heat transfer were investigated, along with the structural characteristics of the sheet. The validation of the numerical results was implemented through experimental data. The results indicate that an increase in Re leads to an increase in circulating flow intensity in the gas-cushion furnace, due to an increase in the peak value of Nusselt number (Nu). However, the variation of Re imposes an insignificant effect on the pressure coefficient distribution over the sheet. At $Re = 24,000$, an upward deformation is evidenced at the middle part of the sheet. As Re decreases to 16,000, relatively slight downward deformation appears at the middle part of the sheet. At an upper/lower distance ratio of 3:5, a uniform pressure distribution is obtained at the lower surface of the sheet. Furthermore, the near-wall flow in the middle part of the sheet brings benefits, and such a layout is responsible for high average temperature of the sheet.

Keywords: gas-cushion furnace; aluminum sheet; deformation; heat transfer; Reynolds number; distance ratio

1. Introduction

Substituting steel with aluminum alloy enables a significant reduction in a vehicle's weight [1]. This shift not only enhances fuel efficiency but also contributes to improved performance and reduced emissions [2]. The use of aluminum alloys offers a compelling alternative to conventional materials, aligning with the growing emphasis on sustainability and energy conservation in the automotive industry [3]. The aluminum alloy has been widely recognized for its low density and high elasticity and plasticity [4]. Moreover, aluminum alloy is corrosion-resistant, easy to process and form, and recyclable. A wide-spread application of aluminum alloy sheets for automotive industry would significantly reduce the vehicle weight, decrease fuel consumption, and lower exhaust emissions [5]. A gas-cushion furnace serves as a crucial heat treatment equipment for alloy sheet processing. Gas is ejected from arrays of nozzles, creating a gas cushion that supports and treats the alloy sheets without physical contact [6]. High-velocity and high-temperature gas impacts the sheet surface perpendicularly or at some inclination angle. The gas flow disturbs the thermal boundary layer attached to the sheet surface, and the convective heat transfer is enhanced [7].

The forced-convection heat transfer is predominant in the gas-cushion furnace where the target sheet is impacted by high-velocity jet streams. With the computational fluid dynamics (CFD) technique, Shah [8] investigated the case featuring the impingement of a moving wall by multiple jets and demonstrated that wall motion alters both flow and heat transfer characteristics. Similarly, Huang et al. [9] simulated the interaction

between a planar jet and a moving plate and concluded that the Nusselt number (Nu) increases with plate speed when the plate motion is consistent with jet development. Chattopadhyay [10] employed the large eddy simulation (LES) method to investigate the impact of rectangular slot jets of high Reynolds number (Re) on a moving plate. They observed that the turbulent kinetic energy initially increases with plate speed but decreases when the latter exceeds some threshold.

Geometric parameter and orientation of nozzles significantly influence the processing quality of the plate in the gas-cushion furnace. Wright et al. [11] investigated the heat transfer when five rows of circular jets impinge a concave wall at different concave angles. They observed that at a concave angle of 90° , overall heat transfer coefficient reaches its maximum, and the swirling flow in the stagnation zone is suppressed. Fénot et al. [12] studied quasi-annular jets and concluded that high blockage ratios promotes the entrainment of ambient air. Talapati et al. [13] impacted a flat plate using inclined circular air jets. Their results showed that the distribution of Nu is asymmetric at small jet angles, and the position corresponding to the maximum Nu deviates from the impingement point. Afroz and Sharif [14] analyzed the heat transfer between twin inclined jets and an isothermally heated wall. It indicated that the averaged Nu is inversely related to the distance between the jet and the target plate. Through simulating rotating channel jets, Li et al. [15] found that Coriolis and centrifugal forces induce jet deflection and accordingly, Nu decreases by 12–25% with increasing rotation number. Liu et al. [16] demonstrated that relative to a single jet, an array of jets can achieve 6.64% higher Nu when the flow rate is certain. Demir and Turgut [17] concluded that heat

transfer in the stagnation zone is enhanced at small nozzle-plate distances.

The experimental work on the gas-cushion furnace has been overwhelmingly conducted in the model scale. Caggese et al. [18] investigated the flow in the stagnation region of impinging jets and observed high velocity difference between air and tracing particles, which causes accumulation of particles in the stagnation region. Senter and Sollicec [19] used the particle image velocimetry (PIV) to measure the flow field when the slot jet impinged a moving plate, showing that flow patterns remained consistent across $Re = 5300\text{--}10,600$ but varied considerably with the jet-plate velocity ratio. Shi et al. [20] used a circumferential array of air jets to impact cylinders, noting that at a jet angle of 45° , the heat transfer in the stagnation zone is the most intensified. Xing and Weigand [21] employed the liquid crystal thermography to optimize the jet array and concluded that dimpled plates outperform flat ones by 12–18% in Nu enhancement. Weng et al. [22] obtained the optimal performance at an impingement angle of 90° . Through varying the curvature of the semicircular concave surface, Talapati and Katti [23] revealed secondary Nu peaks and concluded that low curvature ratio is responsible for high averaged Nu .

While extensive research has been conducted on the heat transfer during the jet impingement, two aspects deserve further consideration. One is the deformation of the suspended sheet impacted by the high-velocity jet streams. This issue is distinct when a thin sheet is processed. The other one is the relationship between heat transfer and confined gas flows. The reason that the obtained conclusions are difficult to generalize lies in insufficient understanding of the thermo-fluid dynamics in the gas-cushion

furnace. Especially, when multiple nozzle units operate in combination, the situation is more complex, while the discussion in this aspect has seldom been reported. Furthermore, an in-depth investigation in the two aspects is necessitated for the practical design and application of the gas-cushion furnace.

The present study aims to elucidate flow and heat transfer characteristics when a suspended aluminum sheet translates between upper and lower nozzle units., and the structural response of the sheet was also investigated. After being validated through a comparison between numerical and experimental results, the CFD scheme was used to simulate flow and heat transfer when Re increases from 8,000 to 32,000. Multiple nozzle units involving nozzles of 10 mm in diameter and 45° in inclination angles were used. Three distance ratios, namely 3:5, 1:1, and 5:3 was adopted. Then the fluid-structure interaction (FSI) method was used to calculation sheet deformation based on pressure distributions over the impacted sheet. A comparison of flow, heat transfer, and sheet deformation was implemented. The obtained conclusions are expected to deepen the knowledge of the heat transfer mechanism in such a scenario and provide a comprehensive support to the design and operation of the gas-cushion furnace.

2. Experimental methodology

2.1 Experimental setup

An experimental rig was designed and fabricated to measure the distribution of Nu over the target sheet impinged by air. A schematic view of such rig is shown in Fig. 1(a). It mainly consists of a centrifugal fan, a nozzle module, a positioning module, and a data

acquisition module. Air discharged from the centrifugal fan is used to impinge the aluminum sheet, which can translate in the x direction. An insulation layer is deployed beneath the sheet, as shown in Fig. 1(b). The operating parameters of each module can be adjusted through the control system.

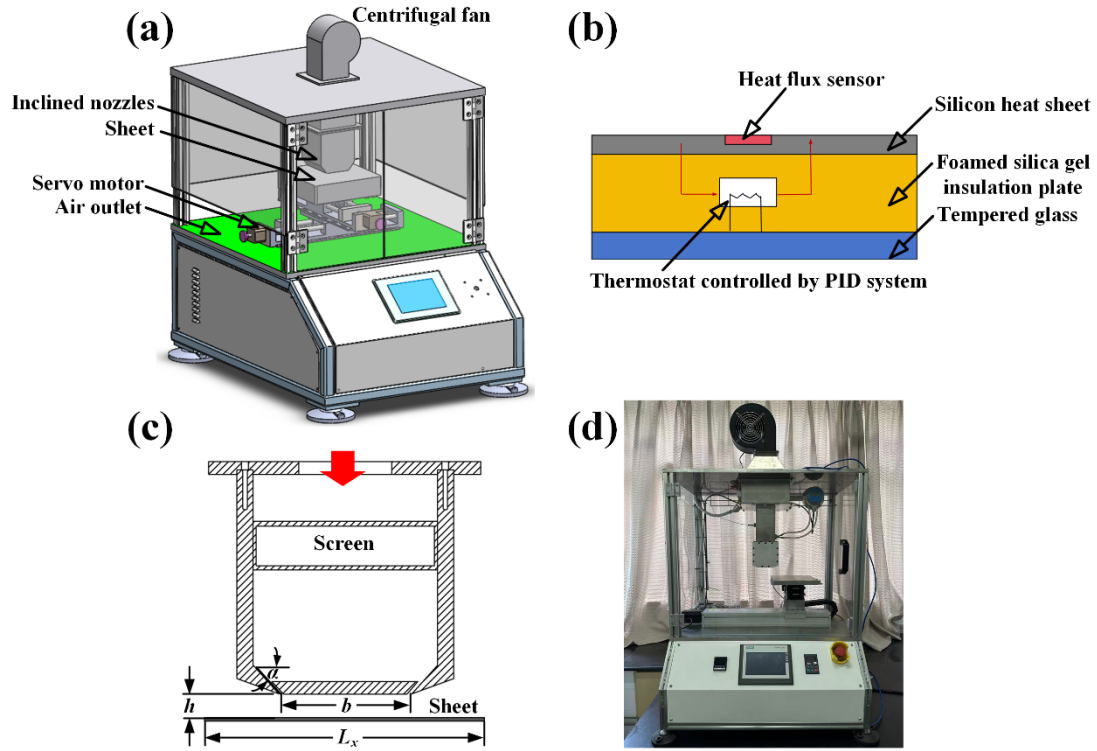


Fig. 1. Experimental rig: (a) Schematic view; (b) Sheet assembly; (c) Cross section of nozzle unit; (d) In-situ image.

2.2 Module parameters

The rated volumetric flow rate and power of the centrifugal fan are $9.5 \text{ m}^3/\text{min}$ and 240 W , respectively. The rotational speed of the fan is adjusted through a variable frequency adjustable-speed equipment. Two heaters of 3 kW rated power are used to heat the air. Air flows through a rectangular duct equipped with rectifying screens, which ensure uniform air flow within the duct.

The nozzle module comprises a $100 \times 100 \times 100 \text{ mm}$ cubic air duct and a dual-slot

nozzle. A cross-section of the nozzle is shown in Fig. 1(c), where the red arrow indicates the incoming air flow. The main dimensions of the nozzle are listed in Table 1. Two rows of air distribution holes are deployed inside the nozzle to ensure uniform velocity distribution at the slot exit. The positioning module includes a 200×200 mm aluminum sheet, a servo motor lift platform, and a horizontal displacement module with a movable range of 400 mm. To adjust the relative orientation of the sheet and nozzle, a multi-axis linear module is applied to position the nozzle with an accuracy of 0.05 mm.

Table 1. Main geometric parameters of the nozzle.

Parameters	Values
Separation distance (h), mm	0–60
Slot width (e), mm	2
Impinging angle (α), °	45
Length of sheet (L_x), mm	200

The data-acquisition module incorporates temperature and airflow rate sensors, a PID control system, and a specifically developed data acquisition and processing software. With this module, air temperature can be precisely regulated. A PT100 platinum resistance temperature detector (RTD) with an accuracy of $\pm 0.15^\circ\text{C}$ was deployed in the airflow channel to measure air temperature under different operating conditions. Another four PT100 RTDs were symmetrically positioned on the aluminum sheet to monitor sheet temperature. A high-precision heat flux sensor (gSKIN XM269C, Switzerland) with dimensions of $4.4 \times 4.4 \times 0.5$ mm was mounted on the aluminum sheet to record instantaneous heat flux during air impingement. An in-situ image of the experimental rig is shown in Fig. 1(d).

2.3 Measurement of Nu

For the heat transfer experiment, Nu was served as a quantitative indicator to evaluate the convective heat transfer efficiency during the jet-sheet impingement process. Since no phase change processes such as evaporation are involved, a simplified form of the wall thermodynamic boundary condition, specifically, a constant temperature boundary condition, is employed. To describe heat transfer characteristics of the wall surface, the data acquired using the heat flux sensor are used to determine the distribution of Nu , which is calculated from:

$$Nu = \frac{qD_h}{k(T_w - T_j)} \quad (1)$$

Where D_h denotes the characteristic length of the nozzle, k represents the thermal conductivity, q the heat flux, and T_w and T_j represent the wall surface and reference temperatures, respectively. Generally, the air temperature at the nozzle outlet is selected as the reference temperature.

The experimental uncertainty was evaluated through the Kline and McClintock method [24]. For temperature measurement, the combination of the accuracy of PT100 RTDs (± 0.15 °C), data acquisition error (± 0.1 °C), and spatial variation (± 0.2 °C) yielded a standard uncertainty of ± 0.27 °C. At a coverage factor $k = 2$, the expanded uncertainty of ± 0.54 °C was obtained. For the calculation of Nu , the primary uncertainty originated from the temperature difference ΔT , resulting in a relative uncertainty of $<5\%$ for typical conditions ($\Delta T \geq 10$ °C). The heat loss accounted for $<1.5\%$ of the total heat input. Therefore, the overall relative uncertainty was $\pm 3.27\%$ for the calculation of Nu . The uncertainty for calculation of Nu is evaluated through:

$$\delta Nu = \sqrt{\left(\frac{\partial Nu}{\partial \Delta T} \delta \Delta T\right)^2 + \left(\frac{\partial Nu}{\partial q} \delta q\right)^2} \quad (2)$$

Where $\delta \Delta T$ and δq are the uncertainties associated with measurements of temperature difference and heat flux, respectively. The accuracy of all measurement probes and experimental uncertainties are listed in Table 2

Table 2. Summary of Measurement Uncertainties

Parameter	Instrument/Calculation	Accuracy	Standard Uncertainty	Expanded Uncertainty (k=2)
Temperature	PT100 RTD	$\pm 0.15 \text{ }^{\circ}\text{C}$	$\pm 0.27 \text{ }^{\circ}\text{C}$	$\pm 0.54 \text{ }^{\circ}\text{C}$
	Signal converter (MIK-403Y)	$\pm 0.1 \text{ }^{\circ}\text{C}$	—	—
	Infrared thermal imager (FLIR A655sc)	$\pm 0.2 \text{ }^{\circ}\text{C}$	—	—
Nu	Derived (ΔT -dependent)	—	$< 5\%$ $\Delta T \geq 10 \text{ }^{\circ}\text{C}$	$\pm 3.27\%$
Heat Loss	Derived	$< 1.5\%$	—	—

3. Numerical model

3.1 Governing equations of gas flow

The air jet discharged from a dual-slot nozzle impinges on the sheet. The fluid dynamics are described by the Reynolds–Averaged Navier–Stokes (RANS) equations, coupled with the energy conservation equation [25]. To achieve closure for these governing equations, the SST $k-\omega$ turbulence model is utilized. This approach ensures accurate representation of turbulent flow behavior and energy transfer within the system. The governing equations describing three-dimensional, incompressible fluid motion are formulated through the conservation principles of mass, momentum, and thermal energy, expressed mathematically as follows:

Continuity equation:

$$\frac{\partial u_j}{\partial x_j} = 0 \quad (3)$$

Momentum equation:

$$\rho \frac{\partial U_i}{\partial t} + \rho U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \rho \overline{u'_i u'_j} \right] \quad (4)$$

Energy equation:

$$\rho \frac{\partial T}{\partial t} + \rho U_j \frac{\partial T}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\frac{\mu}{Pr} \frac{\partial T}{\partial x_j} - \rho \overline{T' u'_j} \right] \quad (5)$$

where U_i , T and P represent the mean velocity, temperature, and pressure, respectively.

T' and u'_i denote the fluctuating components of temperature and velocity. x_i indicates the coordinate directions. ρ is the air density, μ is the dynamic viscosity of air, and Pr is the Prandtl number, which can be obtained from:

$$Pr = \mu \frac{c}{k_f} \quad (6)$$

where c represents the specific heat capacity measured at constant pressure, and k_f denotes the thermal conductivity of air.

The Reynolds stress and turbulent heat flux, which appear in Eqs. (3) and (5), are expressed as functions of variables resolvable on the computational grid.

$$-\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (7)$$

$$-\rho \overline{T' u'_j} = \alpha_t \frac{\partial T}{\partial x_j} \quad (8)$$

where μ_t and α_t correspond to the turbulent dynamic viscosity and turbulent thermal diffusivity, respectively.

The turbulent dynamic viscosity can be expressed as:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (9)$$

where C_μ is a constant, and α_t is connected to the eddy viscosity through the turbulent Prandtl number, Pr_t :

$$Pr_t = \frac{\mu_t}{\rho \alpha_t} \approx 0.9 \quad (10)$$

The intensity of convective heat transfer during air jet impingement on the sheet is evaluated using Nu , which is determined by the thermal resistance characteristics of the aluminum sheet:

$$Nu_x = -\left(\frac{D_h}{T_w - T_j}\right)\left(\frac{\partial T}{\partial n}\right)_{z_{wall}} \quad (11)$$

The characteristic length of the nozzle is represented by D_h , while T_w and T_j correspond to the sheet and air temperatures. The unit vector perpendicular to the sheet's surface is indicated by n .

For the average Nusselt number calculation, the following expression is used:

$$Nu_{avg} = \frac{1}{A} \iint Nu_x dA \quad (12)$$

where A represents the area of the aluminum sheet.

A dimensionless parameter, the pressure coefficient (c_p), is introduced to characterize the pressure distribution, calculated as the ratio of sheet surface static pressure to nozzle outlet dynamic pressure, with its mathematical formulation given by:

$$c_p = \frac{2p_c}{\rho u_j^2} \quad (13)$$

The static pressure at the sheet surface, denoted by p_c , while u_j represents the air velocity at the nozzle outlet. The mean pressure coefficient, $c_{p,avg}$, is determined through the following relation:

$$c_{p,avg} = \frac{1}{A} \iint c_p dA \quad (14)$$

The Reynolds number of the gas jet, Re , is defined as:

$$Re = \frac{\rho u_j D_h}{\mu} \quad (15)$$

The hydraulic diameter, denoted as D_h , is calculated based on the dimension of the nozzle gap.

3.2 Structural relations for the sheet

To describe the structural response of the sheet to the impact of gas streams, the forces acting on the sheet should be analyzed. In a three-dimensional rectangular coordinate system, the differential equation of equilibrium with respect to the element in the sheet can be expressed by [26]:

$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + X_s = 0 \quad (16)$$

$$\frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \frac{\partial \tau_{yx}}{\partial x} + Y_s = 0 \quad (17)$$

$$\frac{\partial \sigma_z}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + Z_s = 0 \quad (18)$$

where τ denotes stress in a plane. Under external loads, the stress at a point in the structure can be expressed through six components, namely σ_x , σ_y , σ_z , τ_{xy} , τ_{xz} and τ_{yz} . X_s , Y_s , and Z_s are components of the body force per unit volume in the x , y and z directions, respectively.

The geometric equation of the sheet takes the following form:

$$\varepsilon_x = \frac{\partial u}{\partial x}, \gamma_{yz} = \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \quad (19)$$

$$\varepsilon_y = \frac{\partial v}{\partial y}, \gamma_{zx} = \frac{\partial u}{\partial z} + \frac{\partial v}{\partial x} \quad (20)$$

$$\varepsilon_z = \frac{\partial w}{\partial z}, \gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \quad (21)$$

where the strain at any point of the structure can be decomposed into normal strain, namely ε_x , ε_y and ε_z , and shear strain, namely γ_{xy} , γ_{xz} , and γ_{yz} .

The constitutive relation is expressed by:

$$\varepsilon_x = \frac{1}{E} [\sigma_x - \nu(\sigma_y + \sigma_z)] \gamma_{yz} = \frac{\tau_{yz}}{\mu_\varepsilon} \quad (22)$$

$$\varepsilon_y = \frac{1}{E} [\sigma_y - \nu(\sigma_z + \sigma_x)] \gamma_{zx} = \frac{\tau_{zx}}{\mu_\varepsilon} \quad (23)$$

$$\varepsilon_z = \frac{1}{E} [\sigma_z - \nu(\sigma_x + \sigma_y)] \gamma_{xy} = \frac{\tau_{xy}}{\mu_\varepsilon} \quad (24)$$

where E denotes the elastic modulus, and μ_e denotes the Poisson's ratio.

The method of FSI is used to study the coupled behavior between gas flow and structural deformation of the sheet. Unlike traditional dynamic grid techniques where the grid motion is predefined, FSI assigns material properties to the solid domain and dynamically updates the fluid and solid grids through exchanging forces and displacements at their interface. This interaction captures complex phenomena arising from the mutual influence of the jet streams and target sheet.

The fluid-solid coupling interface should satisfy the conservation of the fluid and the solid corresponding variables:

$$\vec{\tau}_f \cdot \vec{n}_f = \vec{\tau}_s \cdot \vec{n}_s \quad (25)$$

$$d_f = d_s \quad (26)$$

$$q_f = q_s \quad (27)$$

$$T_f = T_s \quad (28)$$

where τ is stress; d is displacement; q is the heat flux; T is temperature. The subscripts f and s represent fluid and solid phases, respectively.

3.3 Geometric model

The spatial configuration between the aluminum sheet and nozzle array is illustrated in Fig. 2. The aluminum sheet undergoes translational motion in the x direction, at a sheet-to-nozzle velocity ratio between the two arrays of nozzles. The purpose is to achieve forced convective heat transfer between the gas jets discharged from the slot nozzles and the sheet, while maintaining a stable state of suspension for the sheet.

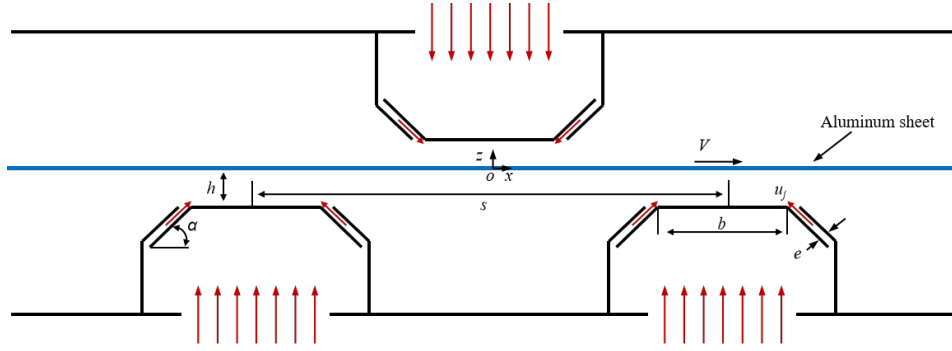


Fig. 2. Schematic view of orientation of the aluminum sheet and nozzle units.

As the thermal processing is initiated in the gas-cushion furnace, heated atmospheric gas is pressurized into the nozzle modules. This process results in the creation of inclined jet streams via the specially designed nozzles, which then collide with the target sheet. Eventually, gas exits through the exhaust outlet. The sheet has a length of 1,600 mm, a width of 450 mm, and a thickness of 0.6 mm. The configuration features a 10.0 mm gap for each slot, with successive nozzle openings separated by 300 mm and installed at a 45° inclination. Each nozzle assembly is positioned 800 mm from its neighboring unit. Regarding the upper nozzle unit, the centerline of the right nozzle intersects with the aluminum sheet at about $x/e = 30$. In this configuration, the parameter h corresponds to the vertical separation between the nozzle's discharge plane and the sheet surface, while u_j denotes the airflow exit velocity. V indicates the translation speed of the sheet. r indicates the proportional relation between sheet speed and jet velocity.

3.4 Numerical setting

The numerical work was carried out using the commercial code ANSYS Fluent 18.0. The results obtained at the moment when the whole sheet entered the heating unit were processed. Based on the operating conditions of the equipment in the factory, the initial temperature of the aluminum sheet is set to 313 K, and the working fluid is air at 923

K. Physical properties of high-temperature air are determined according to its thermophysical properties [27], provided in Table 3. The velocity inlet boundary condition is specified at the inlet of the computational domain, and the pressure outlet boundary condition is set at the outlet. The total pressure is set equal to the ambient pressure. A transient segregated solver is considered in this work. The continuity, momentum, energy, and turbulent transport equations are discretized using the second-order upwind scheme, and the pressure-velocity coupling is solved through the SIMPLE (Semi-Implicit method for pressure-linked equations) method [28].

Table 3. Main thermophysical parameters of air at 923 K.

Parameters	Values
Density (ρ), kg/m ³	0.3822
Specific heat capacity(c), J/kg·K	1126
Dynamic viscosity (μ), kg/m·s	4.119×10^{-5}
Thermal conductivity (k_f), W/m·K	0.0626

In the computational model, the sheet was approximated as a perfectly rigid structure, with its x -direction translation governed by a custom-developed user-defined function (UDF). The dynamic grid technique was employed to progressively modify the grids in the fluid domain in synchronization with sheet motion. Time step selection for the unsteady analysis was optimized based on grid translation velocity and spatial resolution requirements. For example, for the case of $Re = 16,000$, the time step (Δt) was set to 0.00249 s, and 2,677 time steps were covered. Corresponding computational time of 144 hours was consumed with the workstation equipped with 64 CPU cores. Numerical accuracy was maintained by enforcing a convergence criterion of 10^{-5} for all

monitored variables.

3.5 Validation of numerical scheme

Hexahedral structured grids were chosen for the discretization of the computational region. Significant grid density optimization was conducted in both the wall-adjacent zone and the free jet area to ensure precise resolution of the drastic changes in both thermal distribution and flow velocity. The computational domain is shown in Fig. 3.

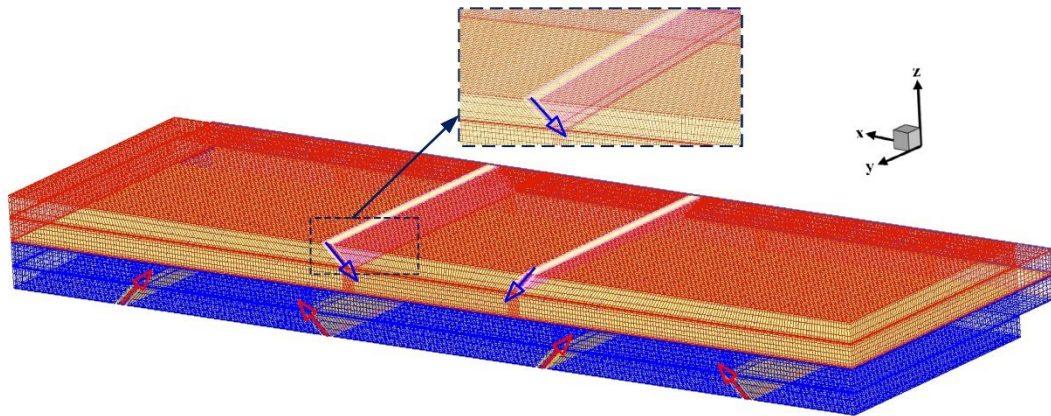


Fig. 3. Computational domain discretized with structured grids.

In order to reduce the influence of the grid on the numerical outcome as much as possible, a grid-independence study was carried out on three grids, totaling 1,762,477, 2,320,579 and 2,815,114 cells, for $h/e = 4$. Numerical simulations were carried out with the same settings at $Re = 10,000$. A comparison of the distribution of Nu along the x -direction on the sheet obtained for all three grids is provided in Fig. 4. It can be noticed that the greatest discrepancy between Grid scheme 2 and Grid scheme 3 is below 1%. Any of those two grids could therefore be used for the simulation work. However, to reduce the computational time, Grid scheme 2 was used in subsequent simulations.

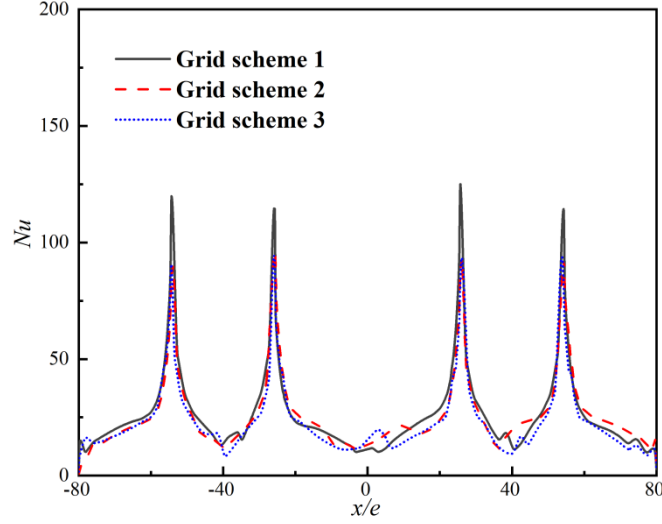


Fig. 4. Distribution of Nu in x direction across different grid schemes.

To match the SST $k-\omega$ turbulence model, the dimensionless parameter y^+ , representing the normalized distance from the wall surface to the initial grid cell, is generally close to 1.0 [29]. The corresponding y^+ on the surface of the sheet in the x direction is shown in Fig. 5. Evaluation of the y^+ distribution shows that while the parameter exceeds 1.0 at the stagnation point, it remains below this threshold in other regions. With all measured values maintaining a maximum limit of 1.5, the computational setup successfully meets the turbulence model's implementation criteria.

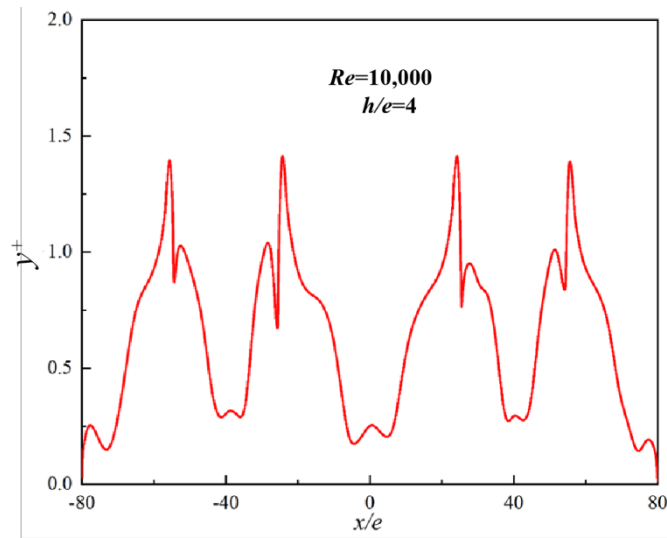


Fig. 5. Variation of y^+ along the x direction for Grid scheme 2.

At $Re = 20,000$, $h/w = 8$, and $R_{pj} = 0.1$, a comparison of the distribution of Nu along the

x direction obtained using three turbulence models, i.e., SST $k-\omega$, v^2-f , and Realizable $k-\varepsilon$ turbulence models, is shown in Fig. 6. The experimental data is presented as well for validation of the numerical scheme. It is observed that the numerical results obtained using the three turbulence models are consistent with the experimental data, especially in the stagnation and intermediate regions, which play an important role during the process of the heat transfer between gas and the sheet. In comparison, the SST $k-\omega$ turbulence model demonstrates higher accuracy of numerical prediction of Nu , and the maximum Nu error is lower than 2 %. For v^2-f and Realizable $k-\varepsilon$ turbulence models, the highest relative deviations from the experimental data reach 9.4 % and 13.4 %, respectively. In consideration of both numerical accuracy and operation characteristics of the gas-cushion furnace, the SST $k-\omega$ turbulence model was selected.

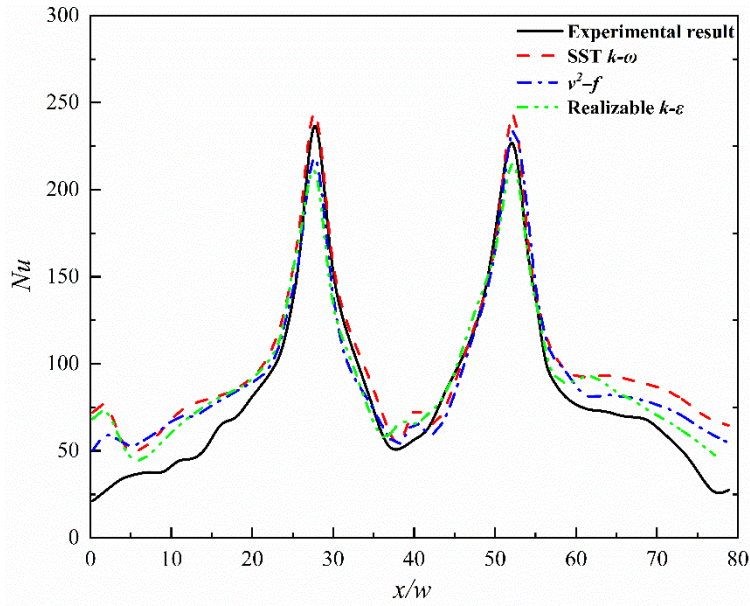


Fig. 6. Comparison of distribution of Nu between numerical results obtained using different turbulence models and experimental result.

The peak values of Nu obtained through experiments are however slightly smaller than those obtained numerically. The discrepancy is explained by the surface roughness of

the sheet, which cannot be accurately predefined in the numerical setup. This will affect the prediction of the convective heat transfer between the air and the sheet [30]. However, there is a significant discrepancy between the numerical data and experimental results on both sides of the sheet. This discrepancy may be attributed to several factors, including surface roughness, the lack of accounting for conduction in the sheet in the simulation, variable fluid flow rates, material property variations, and external disturbances. On the whole, the relative deviation of the numerical data from the experimental results is reasonably low, and the CFD-based results are validated.

4. Results and discussion

4.1 Effects of Re on flow and heat transfer

4.1.1 Flow structure

The flow patterns in $y/e = 0$ at different Re are depicted by Fig. 7, where $h/e = 4$ and $R_{pj} = 0.01$. It can be seen that the flow structures are similar for all Re values considered, however with different intensities. Due to the Coanda effect, the air jets adhere to the sheet, forming wall jets that develop along the sheet surface. The combined effect of the sheet's translational motion and shear forces induces jets detachment from the sheet surface, consequently generating circulation flow patterns.

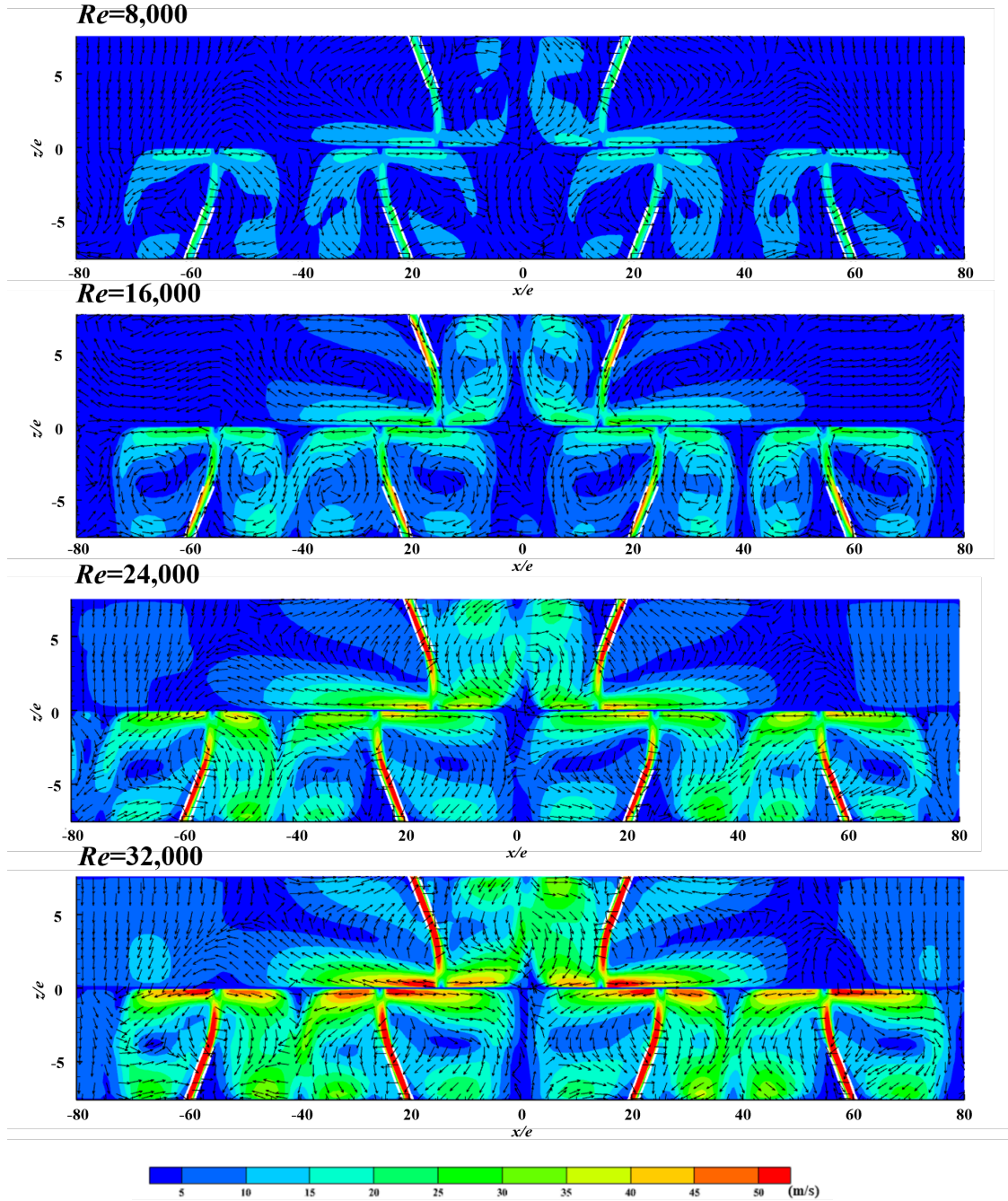


Fig. 7. Velocity vectors distribution at $y/e = 0$ at different Re .

As Re increases, the jet velocity increases significantly, which leads to a corresponding increase in the velocity of the wall jets. Between the two inclined slots of the nozzle unit, the interaction between local flows resulting from the jet impingement on the wall yields two recirculation flows with opposite direction, and secondary stagnation zones are created as a consequence.

Notably, circulation flows also arise in the region between two nozzle units and in the enclosed regions on either side beneath the sheet. The generation of the circulation flows is inseparably related to the constraining effect of the walls and the sheet [31]. When Re exceeds 24,000, high jet velocity leads to the disruption of some vortex structures. The stagnation zones expand, and the circulation flow velocity increases. A comparative analysis reveals that at $Re = 16,000$, the flow pattern is optimum, as it enhances the convective heat transfer between hot air and the sheet.

4.1.2. Temperature and Nu distributions over the sheet

As shown in Fig. 8, the temperature distribution on the sheet surface varies with Re at $h/e = 4$ and $R_{pj} = 0.01$. Due to the sheet motion along the positive x direction, the leading-edge experiences prolonged thermal exposure compared to the trailing part. Therefore, the temperature of the sheet increases progressively from the front to the rear part. Because of the constant velocity ratio between the sheet and the jet, an increase in Re is accompanied by an increase in the velocity of sheet movement, which shortens the heating period of the sheet. As a result, the range of high-temperature areas on the sheet surface is narrowed.

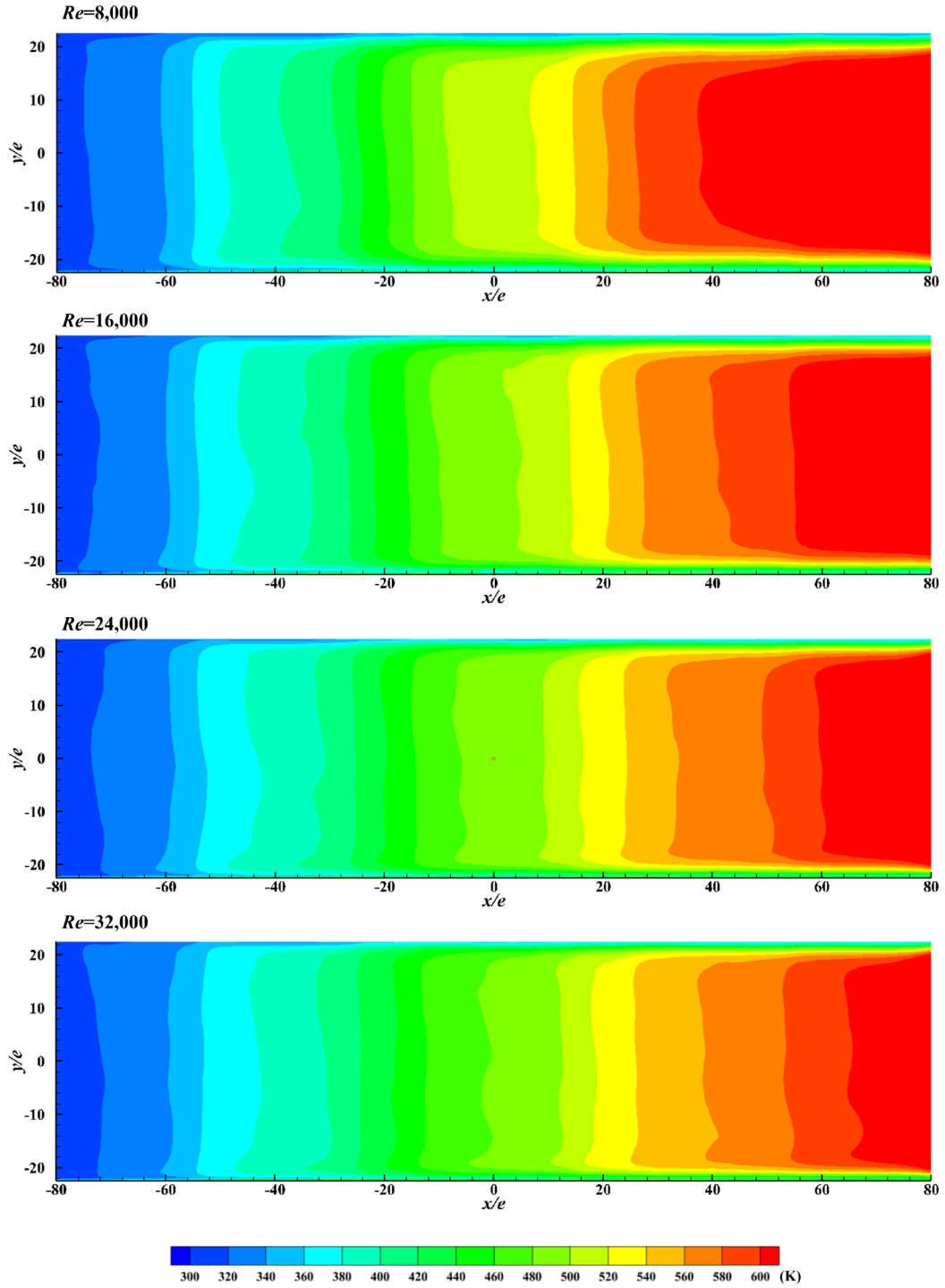


Fig. 8. Distribution of temperature over the sheet at different Re .

The results obtained at $Re = 8,000$ and $16,000$ indicate that at lower Re , the heating time predominantly determines the temperature distribution on the sheet surface. Over this range of variation, the average temperature of the sheet and the temperature gradients decrease as Re values increase. However, at high Reynolds numbers ($Re > 16,000$), the

flow imposes a significant effect on the thermal field on the sheet [32]. Under these conditions, the increase in Re has little effect on average temperature and temperature gradients near the sheet.

The variation of Nu along the x direction at the cross-section $y/e = 0$ is shown in Fig. 9. At the upper or lower sheet surface, the distributions of Nu at different Re are similar. Notably, two distinct Nu peaks are observed at the upper surface, while four pronounced peaks appear at the lower surface, which is related to the number of gas streams. As Re increases, the peak magnitudes rise considerably, leading to an overall enhancement in Nu . This trend is attributed to intensified convective heat transfer due to increased gas velocity in the near-wall zone.

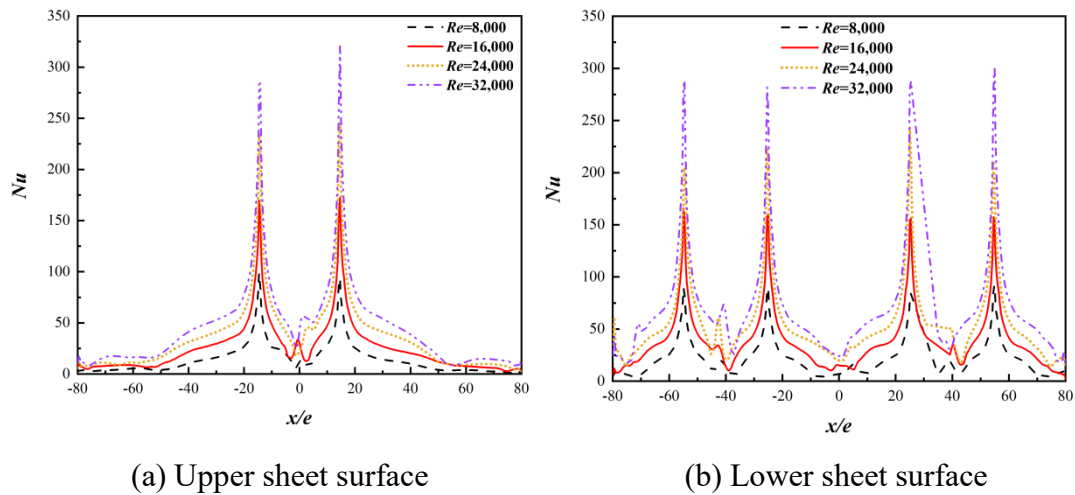


Fig. 9. Distribution of Nu along the x direction at the cross-section $y/e = 0$.

The range of x/e associated with high Nu expands substantially with increasing Re . This is caused by strengthened impingement effect, and the increase in jet velocity promotes the interaction between jet streams and the sheet. Moreover, the crossflow generated with multiple jets further enhances the convective heat transfer in the stagnation zone. With increasing Re , the stagnation zone shifts downstream, i.e., in the x direction. The

deviation from the centerline of the nozzle unit is more pronounced at higher Re . This is related to the synergetic effect of increased sheet speed and shear-induced spent flow, which disturbs the near-wall flow and alters the impingement dynamics.

Furthermore, the distribution of Nu becomes progressively non-uniform with increasing Re , as evidenced by the correspondence between Nu maxima and positions of stagnation zone (see Figs. 7 and 9). This observation demonstrates that jet impingement significantly facilitates heat transfer between gas and the sheet [33]. The inverse relation between uniformity of Nu distribution and Re highlights a critical trade-off, i.e., while high Re improves heat transfer efficiency, it simultaneously reduces thermal homogeneity. Therefore, optimizing operational conditions necessitates a comprehensive consideration of both heat transfer efficiency and uniformity.

4.1.3. Pressure field characteristics over sheet surface

Pressure distributions over the sheet at different Re s are shown in Fig 10, and corresponding distributions of c_p along the x direction over the cross-section $y/e = 0$ are plotted in Fig. 11. As shown in Fig. 10, the highest pressure on both sides arises in the stagnation regions, and there is also a high-pressure area in the secondary stagnation region inside the nozzle unit. The sheet motion in the positive x direction induces shear forces, which result in the displacement of the high-pressure zone within the secondary stagnation area towards the direction of sheet motion. With the increase in the distance away from the nozzle, the pressure gradually decreases and stabilizes at atmospheric levels. On the sheet lower surface, regions of elevated pressure are evident in the central area between the two nozzle units, as well as in the enclosed regions on both sides.

These pressure variations are attributed to the complex interactions among the multiple jets.

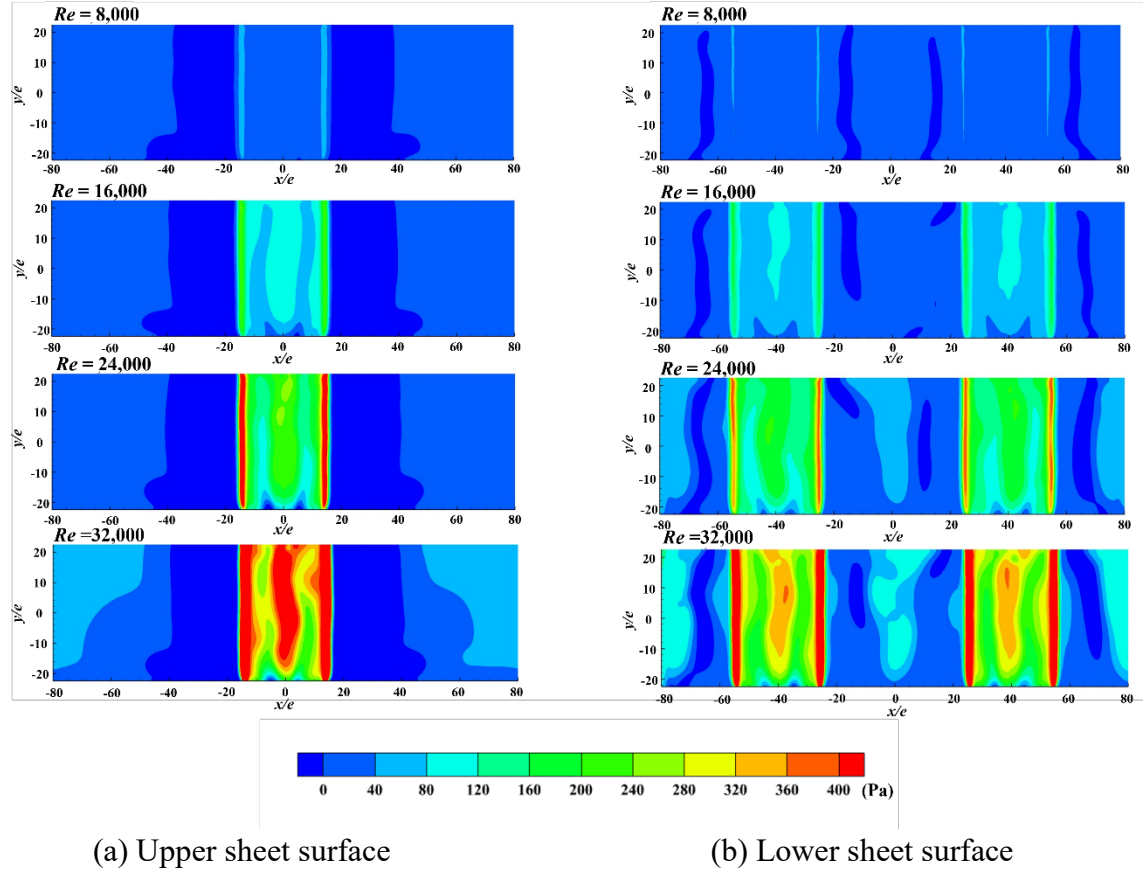


Fig. 10. Pressure distribution on the sheet at different Re .

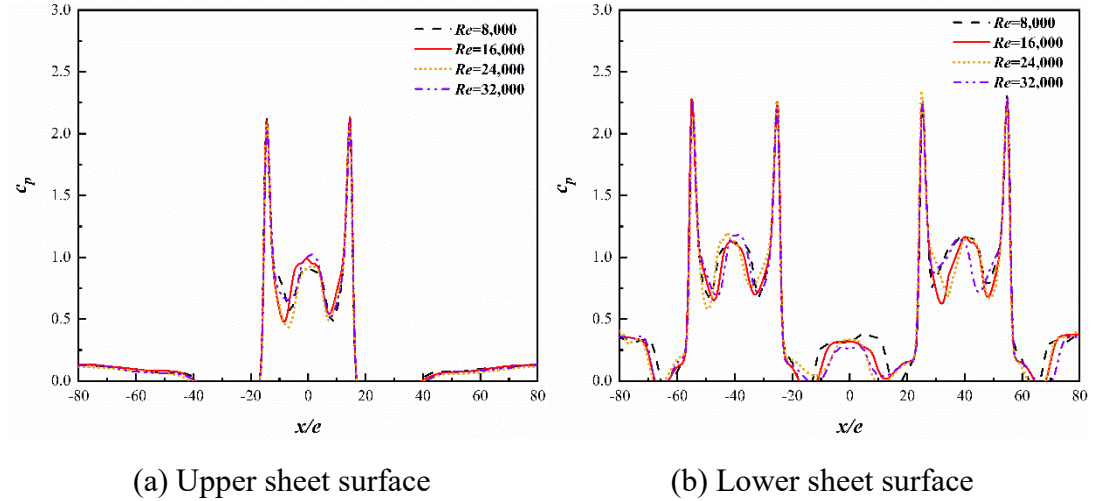


Fig. 11. Distribution of c_p along x direction at $y/e = 0$ at different Re .

An analysis of the velocity vector distribution shown in Fig. 7 reveals that regions of high pressure on the sheet surface correspond to circulation flows. The interaction

between these flow patterns results in pressure increase [34]. The results reported in Ref. [35] further supports the connection between circulation flow patterns and the formation of low-pressure zones. As Re increases, the maximum pressure on both sides of the sheet experiences a substantial rise. However, the sharp increase in pressure within the high-pressure zones leads to a reduction in the overall uniformity of the pressure distribution. As shown in Fig. 11, the distribution of c_p remains largely unaffected by variations in Re , with the peak c_p values staying nearly constant even as Re increases.

4.1.4. Deformation Behavior across the aluminum surface

The vertical displacement profiles (z direction) of the sheet are presented in Fig. 12 for $R_{pj} = 0.01$ and $h/e = 4$, analyzed at $Re = 16,000$ and $24,000$. To account for the tensile forces imposed by the end-mounted rollers, a y -directional fixed boundary condition is implemented in the sheet's computational model. It can be observed from the results that the displacement distribution for the sheet at $Re = 16,000$ and $24,000$, exhibit similar patterns. The regions with apparent deformation are concentrated in the stagnation zone directly impacted by the jet and in the secondary stagnation zone between the two inclined slots. At $Re = 16,000$, the maximum deformation reaches 14.04 mm, with the resulting force acting in the negative z direction. This suggests that the pressure differential is insufficient to counteract the weight of the sheet, leading to significant deformation under the influence of gravity, which in turn impacts the sheet's flatness. The translational motion of the sheet induces asymmetric deformation between its leading and trailing edges. However, when Re is increased to $24,000$, the

deformation pattern changes. Although the significant deformation region remains concentrated in the central area, the resultant force is directed towards a positive z direction. The maximum deformation of the sheet is 18.91 mm, which implies that excessively large pressure difference causes upward sheet deformation. The deformation during the thermal treatment process directly affects the processing quality of the sheet [36]. Comparatively, at $Re = 16,000$, the maximum deformation in the central region is relatively small, and the uniformity of distribution is slightly higher, thus comparative experiments were performed with different distance ratios.

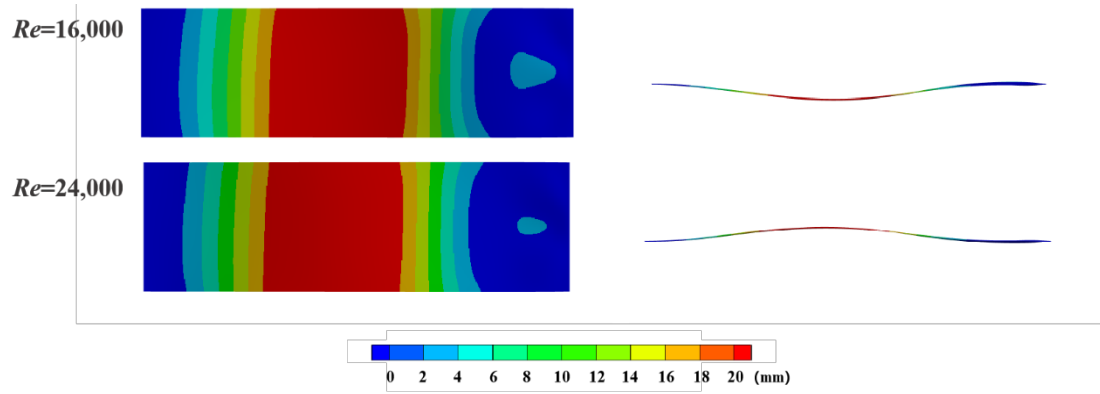
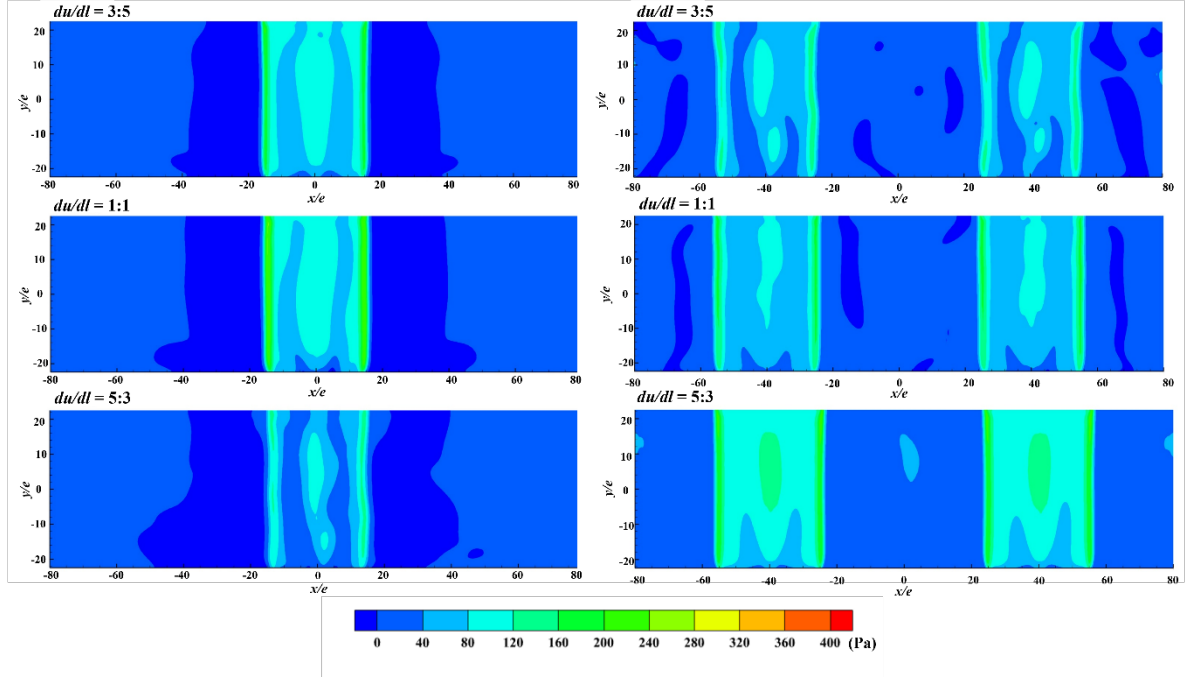


Fig. 12. Top and front view of the distribution of vertical deformation across the aluminum sheet at $Re = 16,000$ and $24,000$.

4.2 Effects of arrangement of the sheet on flow and thermal field

4.2.1. Pressure distribution across the aluminum sheet

The pressure distributions on both sides of the sheet at $Re = 16,000$ and $R_{pj} = 0.01$ are shown in Fig. 13. At different distance ratios (du/dl ; the ratio of the distance from the sheet to the upper nozzle (du), to the distance from the sheet to the lower nozzle(dl)), the highest pressure is produced in the stagnation region of the sheet. Significant pressure elevation is detected not only in the secondary stagnation zones but also in the central area between the two inclined nozzles.



(a) Upper sheet surface

(b) Lower sheet surface

Fig. 13. Pressure distributions on the sheet at different distance ratios and $Re = 16,000$.

When the sheet is arranged at $du/dl = 3:5$, it is relatively close to the upper nozzles, and the pressure on the upper surface is higher. In comparison, the maximum pressure on the lower surface decreases due to enlarged distance between the lower nozzle and the sheet. At $du/dl = 5:3$, the effects are inversed. As evidenced by the deformation pattern shown in Fig. 12, the sheet is sinking in the central region. Therefore, the uniformity of pressure over the lower surface is crucial. At $du/dl = 3:5$, the pressure distribution on the lower surface is more uniform, enabling a large deformation tolerance in the lower region, which promotes stable motion of the aluminum sheet.

The distribution of c_p across the sheet surface is shown in Fig. 14. Variations of c_p obtained at different distance ratios are similar, but differences in the peak values of c_p in the secondary stagnation zone are prominent. These tendencies are associated with circulation flows [37]. The asymmetric arrangement of the sheet alters the furnace space,

causing small circulation flows to merge with larger surrounding flows. This leads to an increase in the peak value of c_p in the secondary stagnation zone and improves the uniformity of pressure distribution in the x direction [38]. Therefore, taking into account the weight and deformation of the aluminum sheet, the distance ratio $du/dl = 3:5$ is responsible for favorable pressure distributions.

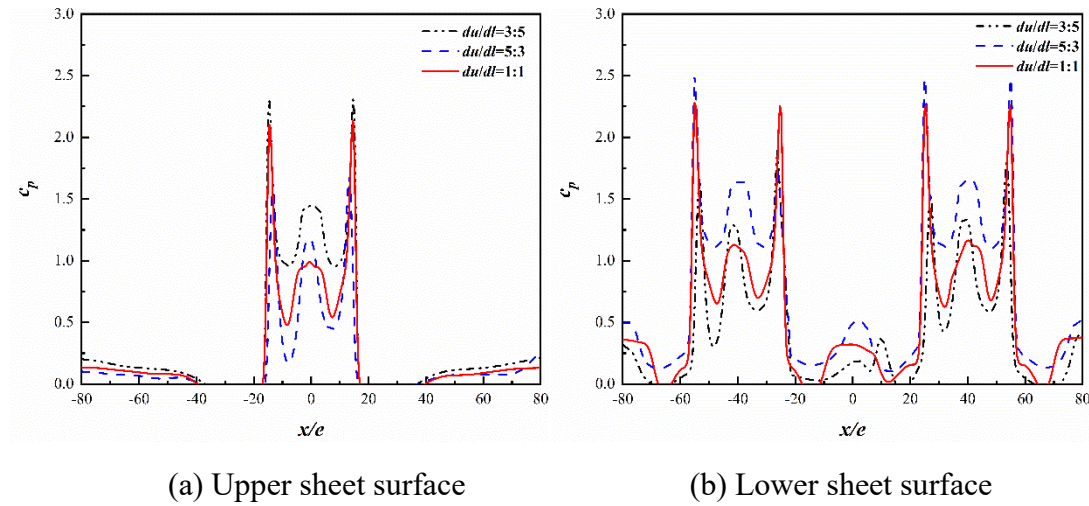


Fig. 14. Distribution c_p along x direction at $y/e = 0$ for $Re = 16,000$.

4.2.2 Thermal field and Nu variation across the sheet

Temperature distributions over sheet surface for $Re = 16,000$ at different distance ratios (du/dl) are shown in Fig. 15. Compared to the symmetric configuration ($du/dl = 1:1$), the case of $du/dl = 3:5$ exhibits higher peak temperatures and increased average temperature. This thermal behavior is attributed to enlarged gap between the sheet and lower nozzles, as creates more space for inter-jet interactions. The expanded spacing promotes vortex merging, leading to the formation of larger-scale vortical structures that intensify gas mixing and consequently improve the convective heat transfer. However, at a distance ratio $du/dl=5:3$, the distribution of temperature on the sheet surface is less uniform, and high temperature gradients are observed.

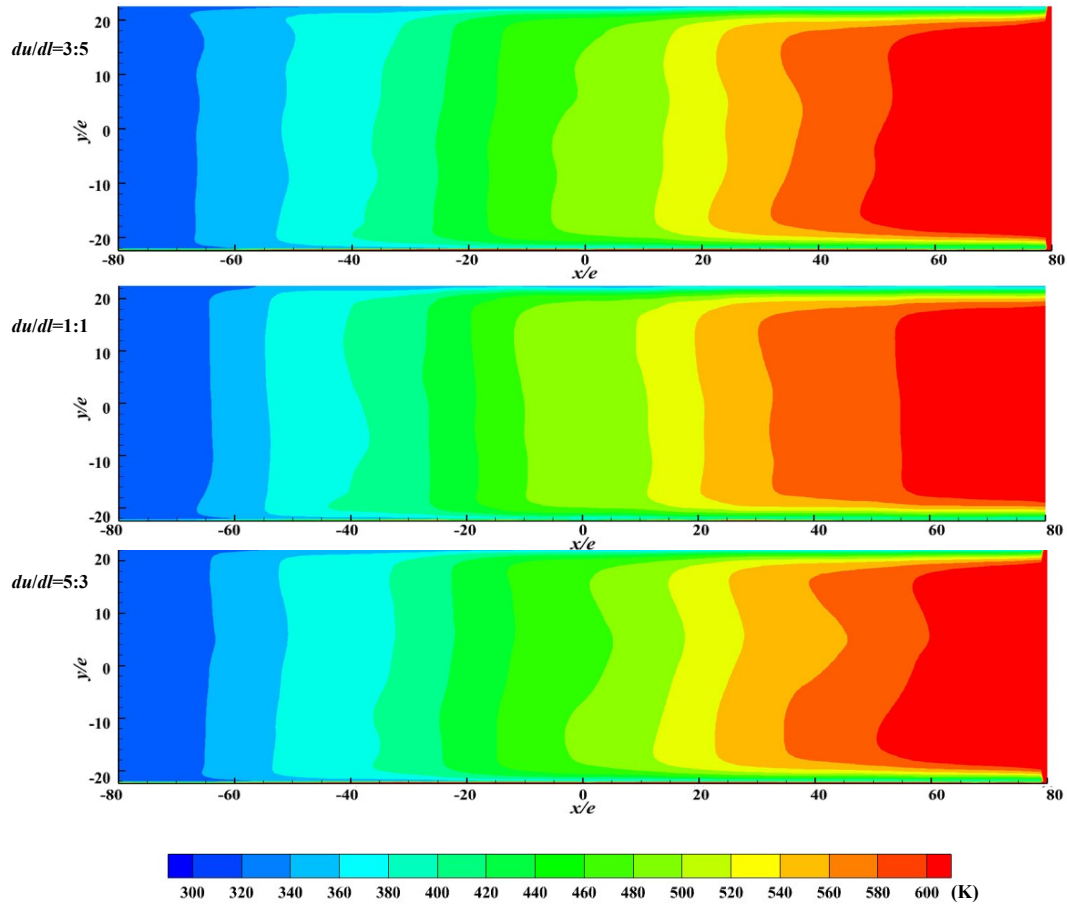


Fig.15. Temperature distributions on sheet surface at different distance ratios and $Re = 16,000$.

At $Re = 16,000$, Nu distributions along the x direction on sheet surface for different distance ratios are shown in Fig. 16. On the upper surface (see Fig. 16(a)), a distinct secondary Nu peak emerges in the secondary stagnation zone at both $du/dl = 3:5$ and $1:1$. In this case, the sheet is closer to the upper nozzles, as enables stronger jet impingement effect. The two upper jet streams impact the sheet at higher velocities, and their interaction creates highly turbulent vortices in the secondary stagnation region, thereby enhancing local heat transfer [39]. In comparison, at $du/dl = 3:5$, both the primary and secondary Nu peaks exhibit higher magnitudes, and their positions shift clearly towards downstream side due to shear-induced effects. In contrast, regarding the lower surface, as seen in Fig. 16(b), no distinct secondary peaks of Nu are identified

at $du/dl = 3:5$ or $5:3$. At $du/dl = 5:3$, the primary peaks of Nu in the stagnation zone are slightly higher than their counterparts, and such a configuration suffers from low uniformity of heat transfer over the sheet.

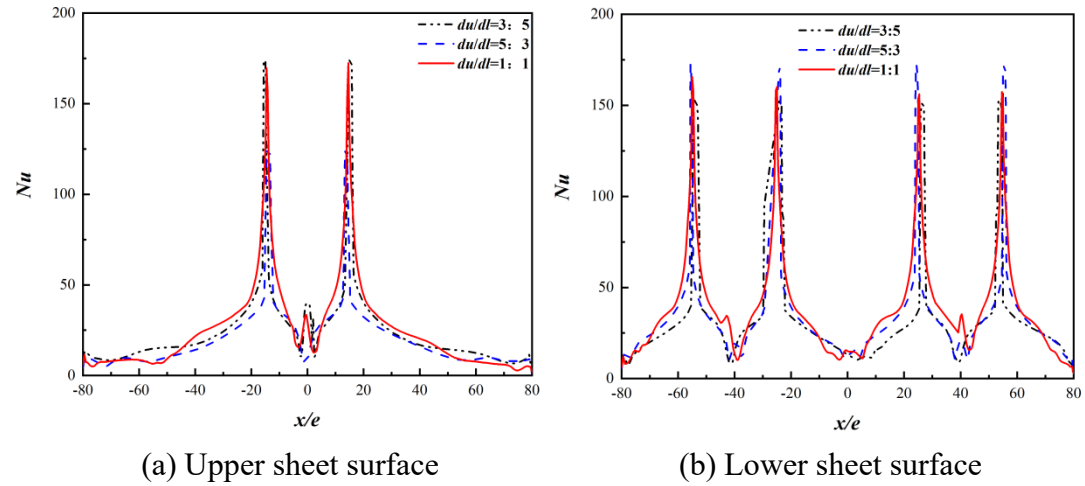


Fig. 16. Distribution of Nu along x direction at different distance ratios for $Re = 16,000$.

4.2.3 Flow structures

The flow patterns in the cross-section $y/e = 0$ at $Re = 16,000$ and different distance ratios are shown in Fig. 17. In these cases, $R_{pj} = 0.01$, and $h/e = 4$. It is demonstrated that the change of the sheet's spatial configuration significantly alters the thermal processing environment in both the upper and lower regions adjacent to the sheet. Consequently, the range of circulation flows in the secondary stagnation zone varies and the symmetrical central area of the nozzles expands considerably in X and Y directions.

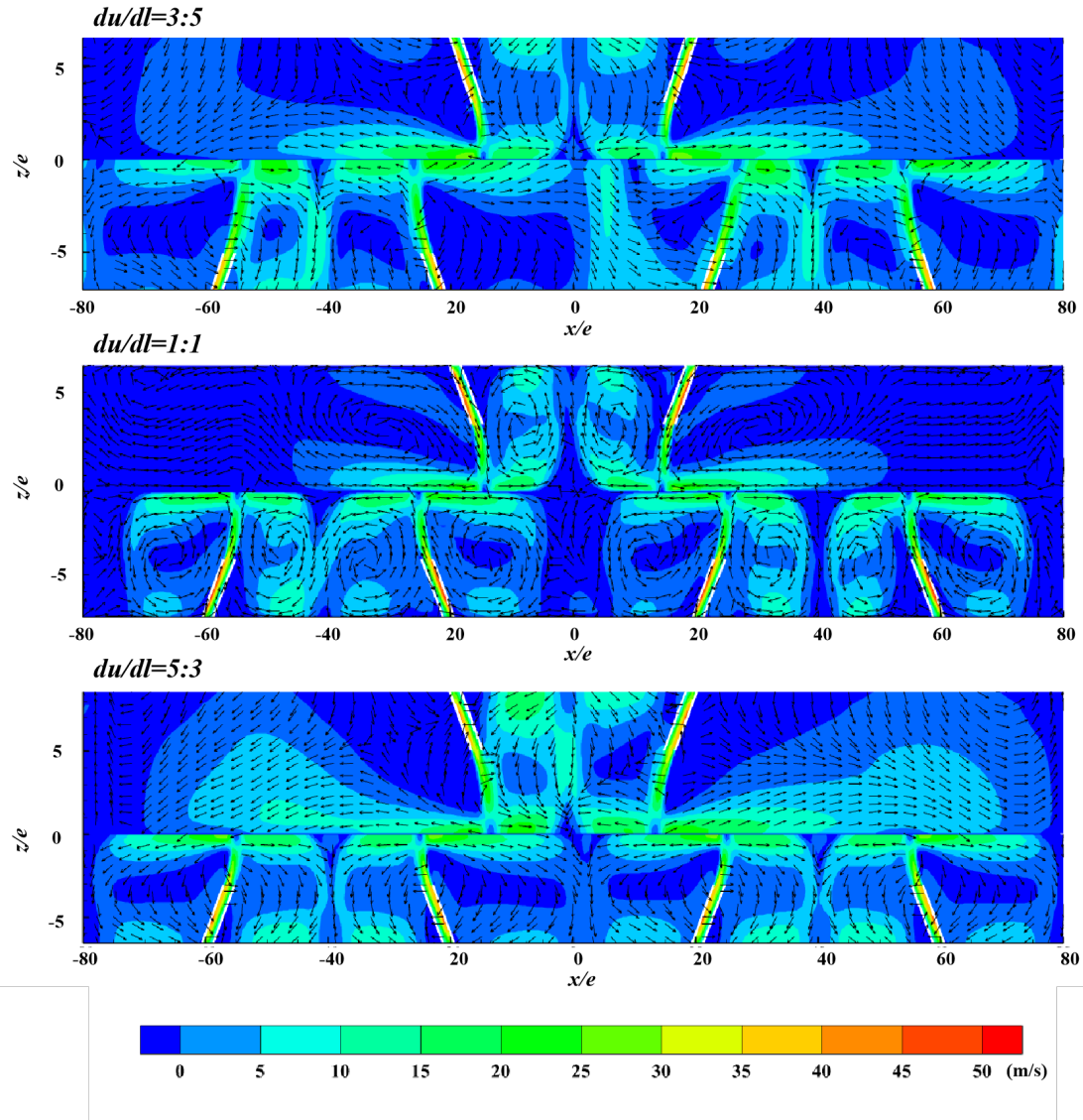


Fig. 17. Distribution of velocity vectors at different distance ratios and $Re = 16,000$.

In the zone below the sheet beneath the nozzle and the closed walls form an enclosure space, the original two circulation flows with opposite direction are merged into a larger one. Distinctly, when the sheet is arranged at $du/dl = 3:5$, a velocity component in the z direction is remarkable for near-wall air at the center of the lower surface of the sheet. This upward velocity component partially counteracts the weight of the sheet facilitating stable suspension of the sheet moving horizontally in the gas-cushion furnace.

5. Conclusions

(1) A significant effect of Re on thermal treatment quality of the aluminum sheet in the gas-cushion furnace is demonstrated. With increasing Re , both the gas velocity and turbulence intensity increase within the jet region. The average temperature of the sheet increases proportionally with Re . High Re is responsible for large peak values of Nu , which causes an improvement of heat transfer efficiency but degradation of the uniformity of heat transfer.

(2) In the primary and secondary stagnation zones, wall pressure is significantly high and overall wall pressure increases with Re . Meanwhile, the distribution pattern of pressure coefficient remains invariant. At $Re = 24,000$, upward deformation of the central part of the sheet in the z direction is clearly evidenced. The deformation is induced by the pressure difference resulting from the joint action of the upper and lower jet streams. At $Re = 16,000$, downward deformation of the middle part of the sheet in the z direction is observed, and the magnitude of deformation is relatively small.

(3) When the sheet is arranged in an upper/lower distance ratio of 3:5, the pressure distribution over the lower surface of the sheet is uniform. Beneath the central region of the lower surface a portion of near-wall air exhibits a velocity component in the positive z direction, and therefore the weight of the sheet is balanced, and stable suspension of the sheet is facilitated. Moreover, at this distance ratio, the high-temperature area on the sheet is large, and the average temperature is relatively high.

Although the present study considers both multiple jet characteristics and structural property of the sheet processed in the gas-cushion furnace, further investigations are

expected to be performed in at least three aspects: (1) Comparison of the response to jet impingement among different materials; (2) Optimization of the nozzle unit and jet pressure or gas flow rate to accomplish high heat transfer efficiency; and (3) Exploration of the relation between turbulent fluctuations and heat transfer with the confined space. Essentially, the three subjects are interrelated, and a comprehensive study will lend a sound support to the flow and heat transfer mechanisms and corresponding engineering applications.

Acknowledgements

The authors are grateful for the financial support from Priority Academic Program Development of Jiangsu Higher Education Institutions of China (PAPD) and Wuxi Science and Technology Development Fund (Grant No. G20212030). The authors also thank for the help in numerical simulation from Key Laboratory of Centrifugal Pumps of China Machinery Industry.

Nomenclature

Greek symbols

μ	Dynamic viscosity of air (kg/m·s)
ν	Kinematic viscosity of air (m ² /s)
ρ	Gas density (kg/m ³)
Ω_{ij}	Angular velocity tensor (1/s)
δ_{ij}	Kronecker delta
α	Inclination angle of nozzle (°)
ε	Turbulence kinetic energy dissipation rate (1/s)
τ	Stress (N/m ²)
μ_ε	Poisson's ratio

List of symbols

A	Area of the heated surface (m ²)
b	Nozzle slot spacing (mm)
c_p	Pressure coefficient
$c_{p,avg}$	Average pressure coefficient
C_μ	Empirical constant
c	Specific heat (kJ/kg·K)
d	Displacement (m)
D_h	Characteristic length of the nozzle (mm)
dl	Distance between lower nozzles and sheet (mm)
du	Distance between upper nozzles and sheet (mm)
E	Elastic modulus (N/m ²)
e	Nozzle slot width (mm)
h	Jet-to-sheet spacing (mm)
k	Turbulent kinetic energy (m ² /s ²)
k_f	Thermal conductivity of gas (W/m·K)
Nu_{avg}	Average Nusselt number
Nu_x	Local Nusselt number
P	Average pressure (N/m ²)
p_c	Wall pressure (N/m ²)
Pr	Prandtl number
q	Heat flux (W/m ²)
Re	Reynolds number
R_{pj}	sheet-to-jet velocity ratio
Sc	Schmidt number
S_{ij}	Angular deformation rate tensor (1/s)
T	Average temperature (K)
T'	Fluctuating temperature components (K)
T_j	Jet temperature (K)

T_w	Wall temperature (K)
t	Time (s)
U_i	Average velocity (m/s)
u_i'	Fluctuating velocity components (m/s)
u_j	Mean jet exit velocity (m/s)
V	Sheet velocity (m/s)
X_s, Y_s, Z_s	Components of the body force per unit volume in the x , y , and z directions (N/m ³)
x, y, z	Coordinate in x , y , z direction (mm)
y^+	Non-dimensional distance between wall and first-layer nodes

Subscripts

f	Fluid phase
s	Solid phase

References

- [1] O. El Fakir, L. Wang, D. Balint, J.P. Dear, J. Lin, T.A. Dean, Numerical study of the solution heat treatment, forming, and in-die quenching (HFQ) process on AA5754, *International Journal of Machine Tools and Manufacture* 87 (2014) 39-48. <https://doi.org/10.1016/j.ijmachtools.2014.07.008>.
- [2] J.C. Benedyk, Aluminum alloys for lightweight automotive structures, in: *materials, design and manufacturing for lightweight vehicles*, Elsevier, 2010: pp. 79–113. <https://doi.org/10.1533/9781845697822.1.79>.
- [3] F. Nturanabo, L.M. Masu, G. Govender, Automotive Light-weighting using aluminium metal matrix composites, *Materials Science Forum* 828-829 (2015) 485-491. <https://doi.org/10.4028/www.scientific.net/MSF.828-829.485>.
- [4] F. Nie, H. Dong, S. Chen, P. Li, L. Wang, Z. Zhao, X. Li, H. Zhang, Microstructure and mechanical properties of pulse MIG welded 6061/A356 aluminum alloy dissimilar butt Joints, *Journal of Materials Science & Technology* 34 (2018) 551-560.

<https://doi.org/10.1016/j.jmst.2016.11.004>.

[5] J. Liu, I. Daigo, D. Panasiuk, P. Dunuwila, K. Hamada, T. Hoshino, Impact of recycling effect in comparative life cycle assessment for materials selection - A case study of light-weighting vehicles, *Journal of Cleaner Production* 349 (2022) 131317. <https://doi.org/10.1016/j.jclepro.2022.131317>.

[6] A.Ü. Tepe, Y. Yetişken, Ü. Uysal, K. Arslan, Experimental and numerical investigation of jet impingement cooling using extended jet holes, *International Journal of Heat and Mass Transfer* 158 (2020) 119945. <https://doi.org/10.1016/j.ijheatmasstransfer.2020.119945>.

[7] H. Keleş, Y. Özmen, Experimental investigation of heat transfer effects in impinging circular jets confined by inclined plates, *International Journal of Heat and Fluid Flow* 108 (2024) 109468. <https://doi.org/10.1016/j.ijheatfluidflow.2024.109468>.

[8] S. Shah, Numerical analysis of heat transfer between multiple jets and flat moving surface, *International Journal of Heat and Mass Transfer* 171 (2021) 121088. <https://doi.org/10.1016/j.ijheatmasstransfer.2021.121088>.

[9] T. Huang, Z. Yang, S. Diao, Z. Huang, Y. Zhang, H. Zhou, Flow structure and heat transfer analysis of the floatation nozzle with a moving wall, *Engineering Computations* 38 (2021) 36–57. <https://doi.org/10.1108/EC-01-2020-0046>.

[10] H. Chattopadhyay, S.K. Saha, Turbulent flow and heat transfer from a slot jet impinging on a moving plate, *International Journal of Heat and Fluid Flow* 24 (2003) 685–697. [https://doi.org/10.1016/S0142-727X\(03\)00062-6](https://doi.org/10.1016/S0142-727X(03)00062-6).

[11] E. Wright, A. Ahmed, Y. Yan, J. Maltson, L.A. Lopez, Experimental and numerical heat transfer investigation of impingement jet nozzle position in concave double-wall cooling structures, *Heat Transfer Engineering* 43 (2022) 1108–1118. <https://doi.org/10.1080/01457632.2021.1943842>.

[12] M. Fénot, E. Dorignac, R. Lantier, Heat transfer and flow structure of a hot annular impinging jet, *International Journal of Thermal Sciences* 170 (2021) 107091. <https://doi.org/10.1016/j.ijthermalsci.2021.107091>.

[13] R.J. Talapati, K. Baghel, H. Shrigondekar, V.V. Katti, Influence of inclined

unconfined circular air jet impingement on local heat transfer characteristics of smooth flat plate, *International Journal of Thermal Sciences* 197 (2024) 108848.
<https://doi.org/10.1016/j.ijthermalsci.2023.108848>.

[14]F. Afroz, M.A.R. Sharif, Numerical study of heat transfer from an isothermally heated flat surface due to turbulent twin oblique confined slot-jet impingement, *International Journal of Thermal Sciences* 74 (2013) 1–13.
<https://doi.org/10.1016/j.ijthermalsci.2013.07.004>.

[15]J. Li, J. Wang, J. Li, Y. Zhao, R. Li, L. Li, Large-Eddy simulation of flow and thermal behavior in jet impingement on a flat plate under rotating conditions, *Int. Commun. Heat Mass Transf.* 156 (2024) 107682.
<https://doi.org/10.1016/j.icheatmasstransfer.2024.107682>.

[16]R. Liu, H. Zhang, M. Nie, L. Yao, X. Li, F. Li, Numerical investigation on the heat transfer performance of the liquid metal array jet impingement, *International Journal of Thermal Sciences* 210 (2025) 109674.
<https://doi.org/10.1016/j.ijthermalsci.2024.109674>.

[17]F. Demir, O. Turgut, Experimental and numerical study of combined impinging jets at low nozzle-plate distance for confined and unconfined impingement plate, *Applied Thermal Engineering* 252 (2024) 123690.
<https://doi.org/10.1016/j.applthermaleng.2024.123690>.

[18]O. Caggese, G. Gnaegi, G. Hannema, A. Terzis, P. Ott, Experimental and numerical investigation of a fully confined impingement round jet, *International Journal of Heat and Mass Transfer* 65 (2013) 873–882.
<https://doi.org/10.1016/j.ijheatmasstransfer.2013.06.043>.

[19]J. Senter, C. Sollicec, Flow field analysis of a turbulent slot air jet impinging on a moving flat surface, *International Journal of Heat and Fluid Flow* 28 (2007) 708–719.
<https://doi.org/10.1016/j.ijheatfluidflow.2006.08.002>.

[20]L. Shi, X. Zhu, Z. Du, Study on convective heat transfer characteristics of inclined jet impinging cylindrical target surface in the confined space, *Applied Thermal Engineering* 218 (2023) 119316.

<https://doi.org/10.1016/j.applthermaleng.2022.119316>.

[21] Y. Xing, B. Weigand, Experimental investigation of impingement heat transfer on a flat and dimpled plate with different crossflow schemes, *International Journal of Heat and Mass Transfer* 53 (2010) 3874–3886. <https://doi.org/10.1016/j.ijheatmasstransfer.2010.05.006>.

[22] J.-Y. Weng, M.-L. Zhan, Y.-H. Liu, Experimental study of single and doublet liquid jet impingement on the liquid film of a horizontal heated plate, *Heat Mass Transfer* 61 (2025) 44. <https://doi.org/10.1007/s00231-025-03566-4>.

[23] R.J. Talapati, V.V. Katti, Local heat transfer characteristics of circular multiple air jet impinging on a semicircular concave surface, *Experimental Heat Transfer* 37 (2024) 851–874. <https://doi.org/10.1080/08916152.2023.2209894>.

[24] R.J. Moffat, Describing the uncertainties in experimental results, *Experimental Thermal and Fluid Science* 1 (1988) 3–17. [https://doi.org/10.1016/0894-1777\(88\)90043-X](https://doi.org/10.1016/0894-1777(88)90043-X).

[25] D. Benmouhoub, A. Mataoui, Inclination of an impinging jet on a moving wall to control the stagnation point location, *International Journal of Thermal Sciences* 89 (2015) 294–304. <https://doi.org/10.1016/j.ijthermalsci.2014.11.007>.

[26] X. Lei, Y. Ma, S. Lin, Numerical simulation on aerodynamic performance of sail wing at different wind speeds based on unidirectional fluid-structure interaction, *Journal of Wuhan Sports University* 53 (11) (2019) 95-100

[27] J. Kim, A numerical study of the effects of ambient wind direction on flow and dispersion in urban street canyons using the RNG $k-\omega$ turbulence model, *Atmospheric Environment* 38 (2004) 3039–3048. <https://doi.org/10.1016/j.atmosenv.2004.02.047>.

[28] S. Patankar, *Numerical Heat Transfer and Fluid Flow*, CRC Press, Boca Raton, (2018). <https://doi.org/10.1201/9781482234213>.

[29] F. Afroz, M.A.R. Sharif, Numerical study of turbulent annular impinging jet flow and heat transfer from a flat surface, *Applied Thermal Engineering* 138 (2018) 154–172. <https://doi.org/10.1016/j.applthermaleng.2018.04.007>.

[30] S. Caliskan, S. Baskaya, T. Calisir, Experimental and numerical investigation of

- geometry effects on multiple impinging air jets, *International Journal of Heat and Mass Transfer* 75 (2014) 685–703. <https://doi.org/10.1016/j.ijheatmasstransfer.2014.04.005>.
- [31] M. Rahimi, R.A. Soran, Slot jet impingement heat transfer for the cases of moving plate and moving nozzle, *Iranian Journal of Science and Technology, Transactions of Mechanical Engineering* 38 (2016) 2651–2659. <https://doi.org/10.1007/s40430-016-0496-8>.
- [32] C. Kang, J. Yin, K. Wang, L. Wu, Effects of Reynolds number and translation speed on flow and heat transfer for aluminum sheet processed in an air-cushion furnace, *Case Studies in Thermal Engineering* 49 (2023) 103271. <https://doi.org/10.1016/j.csite.2023.103271>.
- [33] T. Dairay, V. Fortuné, E. Lamballais, L.-E. Brizzi, Direct numerical simulation of a turbulent jet impinging on a heated wall, *Journal of Fluid Mechanics* 764 (2015) 362–394. <https://doi.org/10.1017/jfm.2014.715>.
- [34] P. Aillaud, F. Duchaine, L.Y.M. Gicquel, S. Didorally, Secondary peak in the Nusselt number distribution of impinging jet flows: A phenomenological analysis, *Physics of Fluids* 28 (2016) 095110. <https://doi.org/10.1063/1.4963687>.
- [35] E. Baydar, Y. Ozmen, An experimental and numerical investigation on a confined impinging air jet at high Reynolds numbers, *Applied Thermal Engineering* 25 (2005) 409–421. <https://doi.org/10.1016/j.applthermaleng.2004.05.016>.
- [36] T. Huang, P. Tan, M. Li, Y. Zhang, H. Zhou, Numerical modeling and analysis of heat transfer for floatation nozzle with a flexible substrate, *International Journal of Thermal Sciences* 137 (2019) 665–674. <https://doi.org/10.1016/j.ijthermalsci.2018.10.031>.
- [37] E. Baydar, Y. Ozmen, An experimental investigation on flow structures of confined and unconfined impinging air jets, *Heat Mass Transfer* 42 (2006) 338–346. <https://doi.org/10.1007/s00231-005-0021-6>.
- [38] A. Hadipour, M. Rajabi Zargarabadi, Heat transfer and flow characteristics of impinging jet on a concave surface at small nozzle to surface distances, *Applied Thermal Engineering* 138 (2018) 534–541.

<https://doi.org/10.1016/j.applthermaleng.2018.04.086>.

[39]M. Rahimi, R.A. Soran, Numerical analysis of heat transfer and flow field due to slot air jet impingement for the cases of moving plate and moving nozzle, Iranian Journal of Science and Technology, Transactions of Mechanical Engineering 41 (2017) 217–224. <https://doi.org/10.1007/s40997-016-0054-0>.



DSPACE

<https://dspace.org/>

Effects of Reynolds number and layout on aerodynamic and heat transfer characteristics of an aluminum sheet treated in a gas-cushion furnace

Wu, Licheng; Kang, Can; Verdin, Patrick G.; Yin, Jin; Xie, Yu

2025-08-01

Elsevier

<https://doi.org/10.1016/j.tsep.2025.103805>

Wu L, Kang C, Verdin PG, et al., (2025) Effects of Reynolds number and layout on aerodynamic and heat transfer characteristics of an aluminum sheet treated in a gas-cushion furnace. Thermal Science and Engineering Progress, Volume 64, August 2025, Article number 103805