A numerical study of the heat transfer of an impinging round-jet methane Bunsen flame

H.S. Zhen⁎, L. Zhang⁎, Z.L. Wei⁎, Z.B. Chen⁎, Z.H. Huang⁎

⁎ The Mechanical and Electrical Engineering College, Hainan University, Haikou, China
⁎ State Key Laboratory of Multiphase Flows in Power Engineering, Xi'an Jiaotong University, Xi'an 710049, China

A R T I C L E   I N F O
Keywords:
Flame impingement
Heat transfer
Combustion modeling
CFD

A B S T R A C T

A numerical model was constructed by Fluent software in this study with the objective to evaluate the model performance of predicting impinging flame heat transfer. The mechanism of methane-air-2step provided by the software and a Chemkin-compatible mechanism are used. With other conditions unchanged, the flame and heat transfer characteristics of the flame were computed and compared.

Both mechanisms over predicted the height and maximum temperature of the flame by referring to the experimental data. The highest temperatures were found in small hot spots which are found to influence the impingement heat transfer at large nozzle-plate distance. Overall, the Chemkin mechanism gives a more reasonable temperature field and better predicts flame height and flame structure due to the more species and reactions considered.

Both mechanisms can follow the experimental heat flux variation. The Chemkin mechanism better predicts heat transfer quantitatively, while the Fluent mechanism may be preferred by engineers for a qualitative analysis with lower computational cost.

1. Introduction

Impinging flame jets are of considerable research interest because of their importance in a wide range of practical applications. For example, impinging flames are preferably used for heating or drying of material in industrial and domestic processes. This method of heat transfer is increasingly used than the more expensive radiant heating technique [1]. The greater heat fluxes obtained with flame impingement shorten the processing time, which ultimately increases productivity, reduces fuel consumption and lowers pollutant emission.

Flame impingement is a complex process in terms of flow-combustion coupling and flow/flame-wall interaction. In the past, lots of studies have been carried out to understand the full picture of the flame impingement system, ranging from the flow and flame structure [2–4], the heat transfer behavior [5–10], to the pollutants emission characteristics [11–13]. Especially, the development of small thermopile sensors, which made localized measurements of heat flux possible, led to a boom of the research on local wall heat flux variations from impinging flames [8–10].

The heat transfer rates of impinging flames are affected by many factors including Reynolds number, fueling rate, jet-to-plate spacing, target plate inclination, roughness of the target plates and etc [1,8]. Although convection is the dominant heat transfer mechanism, conduction, radiation and thermo-chemical heat release may also play important roles [5], which may involve convection, conduction, radiation and thermo-chemical heat release. It is straightforward that a considerable body of the literature has been relied on experiments to investigate the flame impingement phenomenon [2–4,6–13]. The researchers [2,3] utilized a high resolution CCD camera to record the impinging flames at various operating conditions. Makmool et al. [4] adopted the particle image velocimetry (PIV) to obtain detailed information of the impinging flame flow field. For local wall heat flux measurement, both thermal infrared camera [6,7] and heat flux sensor [8–10] have been used. To resolve the pollutants emission problem, the analyzers for combustion product gases, including O2/CO/CO2/NO/NO2 were most commonly used under in-flame probing or out-of-flame probing conditions [11–13].

Compared to the numerous experimental work conducted for impinging flames, the number of numerical studies in literature is rather limited [14–19]. Using the commercial CFD code of Fluent, Chander and Ray [14,15] evaluated the heat flux at stagnation point for laminar impinging methane/air flame. It was found that the reduced mechanism under-predicted the flame length and the heat flux as well. Also focusing on stagnation point, Kleijn [16] presented a simple model

⁎ Corresponding author.
E-mail address: ban18@126.com (H.S. Zhen).

https://doi.org/10.1016/j.fuel.2019.04.077
Received 18 December 2018; Received in revised form 11 March 2019; Accepted 11 April 2019
0016-2361/ © 2019 Elsevier Ltd. All rights reserved.
Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>r</td>
<td>radial distance from flame axis, mm</td>
</tr>
<tr>
<td>z</td>
<td>axial distance from flame axis, mm</td>
</tr>
<tr>
<td>Z</td>
<td>burner-to-plate distance, mm</td>
</tr>
<tr>
<td>Re</td>
<td>jet Reynolds number</td>
</tr>
<tr>
<td>φ</td>
<td>equivalence ratio</td>
</tr>
</tbody>
</table>

A | flame cone surface area, m² |
ρ_u | gas density, kg/m³ |
S_u | unstretched laminar burning velocity, m/s |
δ | flame thickness, mm |
λ | thermal conductivity, W/(m·K) |
C_p | specific heat at constant pressure, J/(kg·K) |
NRR | net reaction rate, kg/(m³·s) |

For heat flow in terms of the flame-tip-to-surface distance. In a study of perforated plate impinging flames, Hindasageri et al. [17] conducted a three-dimensional combustion model to explain the shift in the peak heat flux away from the geometric intended location. A comprehensive numerical study of both stagnation point and radial heat fluxes was carried out by Tajik and Hindasageri [18] for an impinging radial jet reattachment flame. To accurately resolve the heat transfer and pollutants emission characteristics, both detailed and reduced reaction mechanisms were adopted. Comparison of the computed results by various mechanisms showed that the 30 species mechanism was better to predict the heat flux and pressure coefficients while the 49 species mechanism is more accurate for NOx modeling. Jarray et al. [19] performed a transient simulation by Fluent to examine a premixed impinging methane-air flame. The CFD calculations reproduce within 5% of the so-called heat transfer efficiency.

It is clear from the literature review that among the available numerical studies of impinging flames, only a few [14,15,19] examined the single axisymmetric premixed flame jet, though it is the most commonly jet used in practice. Compared to the abundant experimental data obtained, the computational investigation on single circular impinging flame is far from complete. More importantly, there is a need to extend the CFD model from a single flame to multiple flames, as the latter are more commonly used in practical industry. For engineers, the CFD model is an ideal tool to do better burner design. However, the literature shows that the simulation with detailed or reduced mechanism for a single flame is still time-consuming, not to mention the computation cost for a flame array. Before development of an efficient CFD model that can be used for predicting flame array impingement heat transfer with good accuracy, it is necessary to compare the performances of two CFD models, one with a very simple mechanism (6 species and 3 reactions) for reducing the computation cost and another with detailed chemical reactions (30 species and 184 reactions) for higher prediction accuracy, based on a single flame. This paper acts a basic step taken to understand the two coins of the CFD model developed and may provide insights for further improvement of the model feasible for a flame array.

2. Numerical simulation

2.1. Computational model

For flame impingement heat transfer, a 2-D model is solved by Ansys Fluent software (version 16.0). The physical model is that of Ref. [20]. A round pipe with 13 mm inner diameter provides a premixed methane/air flame which impinges vertically normal to a copper plate. Fully developed laminar flow is established due to the large length of the pipe. Thus, a parabolic velocity profile is adopted for the CFD simulation. Due to axisymmetry of the impingement system, only a half domain was used. A schematic for the computational domain is given in Fig. 1 where a much sparser mesh than the actual ones is shown for demonstration and clarity purpose. The radial size of the domain is r = 100 mm which is sufficiently large to allow radial development of the flow. The axial size is changed to model the impinging flame under different burner-to-plate distances, Z. That means for each varied Z, the grid is re-meshed. To better resolve the reactive flow and boundary layer, non-uniform quadrilateral meshes are built. For example, the mesh dimensions are Δr = 300 μm and Δz = 200 μm in the central-flame region within r ≤ 10 mm, while in the near-plate region the axial dimension Δz is decreased by scaling from 1 to 0.25 towards the plate. Outside the central-flame and near-plate regions, the mesh dimension is gradually increased to minimize the total mesh size.

Numerical results are obtained by solving the Navier-Stokes equations in steady-state mode. For all models, the segregated solver with implicit method of advancing toward the solution is used, and the volumetric reactions with stiff chemistry solver is adopted. As the flames are laminar, laminar finite rate turbulence-chemistry interaction is utilized. For modeling methane-air combustion, the 2-step reaction mechanism available in Fluent, i.e. methane-air-2step with 6 species and 3 reactions, is chosen to minimize the computation time. In addition, a Chemkin mechanism of Lu & Law [21] with 30 species and 184 reactions is also used. It is much more time-consuming than Fluent mechanism. This Chemkin mechanism is validated to be an accurate substitute for GRI 3.0 Mechanism by Lu and Law [21], and thus is popularly used for heat flux modeling [17,18,22].

For all computation processes, the SIMPLE algorithm is selected for pressure interpolation and coupling between pressure and velocity. The second-order upwind scheme is adopted for the spatial discretization. The computation is initially started by solving a cold flow. After around 2000 iterations, a small region above the nozzle is patched by an artificially assigned temperature of 2000 K. Then, the solution is further iterated by solving the energy and species equations. For convergence, an under-relaxation factor of 0.1 is initially assigned to all equations except energy equation with a factor of 0.05. Once the flame is ignited, the under-relaxation factors are gradually increased to their default values. Typically, a total number of 20–30 thousand iterations are conducted to achieve a converged result where a residual error of 10^{-6} for continuity, momentum and species equations, and of 10^{-5} for energy equation is met.

![Fig. 1. A qualitative illustration of the computational grid.](image-url)
2.2. Model validation

A total of five grids of varying mesh number are used for grid independence check. For each mesh, there are a total of \( N_r \times N_z \) mesh points in \( r \) and \( z \) direction, respectively. For the possible flame zone at \( 0 \text{ mm} \leq r \leq 10 \text{ mm} \), there are \( n_r \times n_z \) mesh points. To vary the mesh size, both \( N_r \times N_z \) and \( n_r \times n_z \) are increased such that the total number of meshes increases from 14,400 to 72,000, corresponding to Grid 1–5, as shown in Table 1. The five grids are computed firstly by using Fluent mechanism to achieve grid independence.

To validate the numerical model, a baseline premixed methane-air flame impinging upwards to a horizontal plate is firstly studied. Issuing from a round pipe of 13 mm inner diameter, the methane-air jet has a Reynolds number of \( Re = 1500 \), equivalence ratio of \( \Phi = 1.0 \) and burner-to-plate distance of \( H = 42 \text{ mm} \). The Reynolds number is based on the fully developed flow of unburned reactants, and thus the flame is laminar. For flame impingement, the flame height plays a key role for the resultant heat transfer [1,6,8,10,14–16]. Therefore, the data values of flame height as computed from five grids are compared in Table 2. Generally, the height of the reaction cone is defined as the characteristic flame height, which can be obtained by experimentally measuring the axial distance from the burner exit to the flame tip [10,20]. Similar measurement is conducted numerically in this study. As the predicted reaction cone is a thin flame layer, the outer boundary of the layer is used to locate the flame tip. The resultant flame heights, as shown in Table 2 show a monotonic decrease and the decrease rate also monotonically drops with increasing mesh size from Grid 1 to 5. On the other hand, the variation in maximum flame temperature between neighboring grids also becomes lesser from Grid 1 to Grid 5. Note that using Chemkin software, the adiabatic flame temperatures given by the Fluent and Chemkin mechanism is 2280 K and 2230 K, respectively. These results show that a fair agreement in flame height is obtained between Grid 4 and 5.

The local heat flux from the impinging flame to the plate can be directly computed by Fluent software according to Fourier’s law of conduction [17]. As heat transfer is a major concern of the modeling, the mesh independency is further inspected in terms of local heat flux distribution. The local heat fluxes starting from the stagnation point and along the radial direction are obtained and shown in Fig. 2. It is seen that except Grid 1, the other four grids have a good consistency in local heat flux at a radial position of \( r > 20 \text{ mm} \) which is the wall jet region by referring to Fig. 3. While noticeable differences are seen in the stagnation region of \( r < 20 \text{ mm} \). Within the stagnation region, the enlarged graph in Fig. 2 clearly indicates that the local heat flux decays with increasing mesh size from Grid 1 to Grid 5, and the decay rate decreases against increasing mesh size. Therefore, the curves by Grid 3, 4 and 5 are very close to each other in comparison to those by Grid 1 and 2, which indicates that a rather good independency of the grids is obtained between Grid 3, 4 and 5. By referring to the similar flame heights and temperatures yielded by Grid 4 and Grid 5 in Table 2, Grid 4 is adopted for all later computations in this study.

3. Results and discussions

All simulations are conducted on a computational station: Lenovo P710: Xeon E5-2630V4, 2.2 GHz; 10 Cores; 64G Memory. The typical running time is around 10 and 25 h by using Fluent and Chemkin mechanisms, respectively. It means that the computations cost is more than doubled when Fluent mechanism is changed into Chemkin mechanism. As known, the choice of an appropriate reaction mechanism for practical combustion modeling is to make a balance between the simulation accuracy and the computation cost. Therefore, the simulation accuracies of Fluent and Chemkin mechanisms are firstly compared in this study, and the comparison is made between the numerical results obtained for the baseline flame under \( Re = 1500 \), \( \Phi = 1.0 \) and \( H = 42 \text{ mm} \). For briefness purpose, the flames computed by using Fluent and Chemkin mechanisms are sometimes referred to as Fluent flame and Chemkin flame, respectively.

3.1. Flame height

With a parabolic velocity profile at the nozzle exit and when the tip is closed, the flame sheet is conical but not perfectly straight-sided [23]. A flame sheet is a source of heat and a sink of the reactants, characterized by sharp peaks of chemical reaction rates [24]. As aforementioned, the height of the conical sheet is defined as the flame height in both the experimental study of Ref. [20] and this numerical study. Note that the uncertainty in the flame height measured is reported to be as low as 1.5% [20], thus it is appropriate to use this experimental flame height to validate the numerical model in this study.

As \( CH_4 \) is the fuel in this study and it exits as a species in both Fluent and Chemkin mechanisms, thus its peak of heat reaction rate (NRR) is used as an indicator to identify the flame sheet. The resultant flame cones are illustrated in Fig. 3. It shows that Chemkin mechanism predicts a reaction cone shorter than that computed by Fluent mechanism, which are 36.73 mm and 39.73 mm respectively as given in Table 2. Also given in Table 2 is the experimentally measured cone height in Ref. [20] where the baseline flame in an open environment is 35.00 mm high. With the impingement plate, the flame height would further decreases to be lower than 35.00 mm due to the compression effect of the plate [13]. Therefore, in terms of flame height, Chemkin mechanism exhibits a more accurate prediction than Fluent mechanism.

3.2. Flame structure

Besides flame height, there are several other points revealed by Fig. 3. One of them is that the flame sheet of the Fluent flame has more red colors than the Chemkin flame, indicating that the Fluent flame has higher NRR peak values for the \( CH_4 \) fuel. This is straightforward since non-reversible reactions are handled by Fluent mechanism, and in contrast Chemkin mechanism manages reversible reactions. Another point is that in comparison to the straight flame sides and tip as well, the flame root near the burner rim has a much reduced NRR of \( CH_4 \) due to heat loss occurring thereunto. It is expected that at fuel-rich equivalence ratio, fuel penetration across the flame cone will occur in this region.

The last point Fig. 3 reveals is that as the Fluent flame has a higher cone height, its average flame speed is lower than the Chemkin flame. The detailed reason is as follows. First, all \( CH_4 \) fuel is assumed to be burnt out within the flame sheet since the stoichiometric fuel and air are perfectly mixed before burning. In fact, Fig. 3 has hinted a nearly zero value of \( CH_4 \) NNR outside the flame sheet. Second, according to the constant mass burning flux [23–25], the following equation holds between two cases by adopting different mechanisms:

\[
A \rho_0 \dot{S}_{ch} = A_2 \rho_2 \dot{S}_{ch}
\]

where \( A \) is area of the cone surface on the reactant side, \( \rho_0 \) is density of the reactant mixture, \( \dot{S}_{ch} \) is flame propagation speed averaged over the cone surface, and 1 and 2 denote different mechanisms used. Consequently, a larger cone surface area associates with a smaller flame speed.

It is well known that a Bunsen flame has a negative stretch which

<table>
<thead>
<tr>
<th>Grid</th>
<th>( N_r )</th>
<th>( N_z )</th>
<th>( n_r )</th>
<th>( n_z )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>120</td>
<td>120</td>
<td>75</td>
<td>70</td>
</tr>
<tr>
<td>2</td>
<td>300</td>
<td>120</td>
<td>200</td>
<td>70</td>
</tr>
<tr>
<td>3</td>
<td>300</td>
<td>180</td>
<td>200</td>
<td>105</td>
</tr>
<tr>
<td>4</td>
<td>300</td>
<td>200</td>
<td>800</td>
<td>120</td>
</tr>
<tr>
<td>5</td>
<td>300</td>
<td>240</td>
<td>200</td>
<td>140</td>
</tr>
</tbody>
</table>
becomes gradually stronger from the flame root to tip, and thus the stretch rate is very small for most segments of the cone on the shoulder sections [26]. An expression for the stretch rate distribution along the cone surface is proposed in Ref. [27], which shows that the zero stretch occurs near the CH4 NRR peak. Thus, the local value of  \( S_u \) at 2/3-radius radial position tends to be unstretched. Fig. 4 presents the flame structure across the flame sheet. In the figure, the symbol  \( S_u \) stands for component of local flow velocity normal to the flame sheet. Near the flame axis is the potential core of the flame jet, and thus the distribution of  \( S_u \) is parabolic and decreases with  \( r \). Across the flame sheet,  \( S_u \) sharply rises due to thermal expansion and thus the minimum value of  \( S_u \) represents the local flame speed. As shown, the minima are 43.5 cm/s and 48.7 cm/s for the Fluent and Chemkin flame, respectively. This lower local flame speed is consistent with the aforementioned lower average flame speed predicted by using Fluent mechanism. As a result of lower flame speed, the flame sheet is thinner. The thinner flame sheet is also observable from the narrower CH4 NRR peak and from the fact that steep variations in species concentrations occur near the CH4 NRR peak.

Table 2

<table>
<thead>
<tr>
<th>Reaction mechanism</th>
<th>Grid</th>
<th>Flame Height (mm)</th>
<th>Peak Flame Temperature (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluent</td>
<td>1</td>
<td>41.77</td>
<td>2106</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>39.95</td>
<td>2193</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>39.84</td>
<td>2241</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>39.73</td>
<td>2247</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>39.72</td>
<td>2248</td>
</tr>
<tr>
<td>Chemkin</td>
<td>3</td>
<td>36.73</td>
<td>2203</td>
</tr>
<tr>
<td>Zhen et al. [20]</td>
<td>–</td>
<td>35.00</td>
<td>1828</td>
</tr>
</tbody>
</table>

3.3. Flame temperature

Fig. 5 shows the colorized contours of flame temperature obtained by using Fluent and Chemkin mechanisms, respectively. From the figure, the characteristics of the flame temperature distribution can be clearly identified. Firstly, the bulk flame by Chemkin mechanism has more red colors which indicates higher temperatures than the Fluent flame. The higher temperate is obviously due to the presence of exothermic reactions beyond the reaction sheet of the Chemkin flame, as indicated by Fig. 4. Second, there are hot spots in both flames. For the Fluent flame, the hot spot occurs in a region above the tip of the reaction cone, while for the Chemkin flame, the hot spot resides off the tip. Due to the proximity of these hot spots to the impingement plate, it is expected that these hot spots would distinctly influence the heat transfer behavior of the impinging flame.

It is of importance to find the reason for occurrence of these hot spots. Magnified distributions of CH4 NRR near the flame tip are shown in Fig. 3(b). Different from the full picture in Fig. 3(a), the magnified picture has a full range colorbar and thus a closer examination of the CH4 NRR distribution near the tip can be made. Fig. 3(b) illustrates that the CH4 NRR value is higher towards the tip, and thus focusing heat at the tip of the Fluent flame, where a hot spot resides. The CH4 NRR distribution is rather uniform along the cone edges of the Chemkin flame, and this agrees with the experimental data of Ref. [30] presenting that a stoichiometric methane Bunsen flame has a rather constant temperature along the conical surfaces. Further, the CH4 NRR is relatively weak at the tip, and the reason is supposed to be the effect of curvature, which reduced the tip burning intensity at stoichiometric or lean conditions [31].

To explore the reason of hot spot in the Chemkin flame, it is viable to inspect the distribution of OH which is the most abundant of all radicals at stoichiometric or lean conditions [32]. From the numerical result, the OH concentration peak is found to occur at a shoulder position and outside the reaction cone. For clarity, a slightly magnified picture of colorized contours of OH mass fraction is shown in Fig. 6. Also shown in the figure are the streamlines of the laminar flow. Thus, it can be concluded that a large amount of OH radicals are produced behind the flame sheet, and then are conveyed downstream by the flow. Due to chemical reactions in the post-cone region of the flame, especially the reaction between CO and OH, i.e. OH + CO → CO2 + H which is the principal exothermic reaction for oxidation of carbon monoxide into carbon dioxide, a higher temperature is expected further downstream of the OH concentration peak. Eventually, a hot spot is formed at a location off the flame tip.

3.4. Impingement heat flux

Fig. 7 presents the radial heat flux profiles on the impingement plate within a surface area of 0 mm ≤  \( r \) ≤ 100 mm for the baseline flame operating at  \( Re = 1500, \Phi = 1.0 \) and  \( H = 42 \) mm. Also provided in the figure are the experimental data in our previous study [20], which has an uncertainty of 0.7%. Detailed analysis and explanation for such distributions of heat flux can be referred to Ref. [20]. In current study, the focus is laid on the comparison of the numerical and experimental heat flux profiles so as to better amend the computation model.
It is observed that both models using Fluent and Chemkin mechanisms underestimate the heat fluxes in a region spanning $r < 3 \text{ mm}$, and overestimate the heat fluxes at $r > 30 \text{ mm}$. At the region near the stagnation point of $r < 3 \text{ mm}$, the heat fluxes are the highest by Fluent mechanism, and the lowest for the Chemkin mechanism, which results from the higher temperature (hot spot) at the tip of the Fluent flame.

Compared to the Fluent flame, the heat fluxes in the Chemkin flame are higher in the region of $3 \text{ mm} < r < 44 \text{ mm}$. It is the outer layer beyond the flame sheet that touches the impingement plate at $H = 42 \text{ mm}$, see Fig. 5. Thus the higher temperature in the outer layer and especially the hot spot of the Chemkin flame enhances the heat fluxes. The reason that both simulated flames show heat fluxes lower than the experimental ones in this region is probably twofold. First, no radiation effect is taken into account and second, no turbulence is modeled in this study. The hot species of H$_2$O and CO$_2$ are good radiators and their presence near the hot spot of the Chemkin flame may further promote heat transfer [33]. In addition, a turbulence model may improve the heat flux prediction at the location where the flame jet impinges the plate because a high turbulence might be generated thereunto [34].

Beyond $r = 30 \text{ mm}$, the experimental heat fluxes are lower than those predicted using either Fluent or Chemkin mechanism. This might be due to the difference between the physical system and the numerical model. For experimenting, the target plate is a copper plate of 10 mm thickness. Compared to its thickness, the plate has a much larger surface size of 600 mm × 600 mm in order to make the heat transfer along the plate surface negligible compared to that across the plate thickness. Since the copper plate is water-cooled on its upper surface, a temperature gradient is established inside the plate solid. However for modeling, no solid domain is considered for meshing and a non-slip wall boundary condition with 10 mm thickness stands for the heat transfer in the study.

Fig. 3. Reaction cones of the baseline flame as identified by using CH$_4$ net reaction rate (NRR) with Fluent mechanism (left) and Chemkin mechanism (right).
impingement plate. Therefore, that the temperature of the target surface between the physical system and numerical model is kept at the same temperature actually is not the case. In fact, it is the upper surface of the copper plate that is taken into account by the model. In the real system and at large radial position, the temperature difference between the flame and plate becomes small compared to that at small radial position where flame temperature is high. Thus, the temperature variation across the plate thickness can have significant influence on the
3.4.1. Effect of $H$

So far, the numerical data has shown that for both the flame combustion itself and the impingement heat transfer, the 30-species-184-reactions Chemkin mechanism behaves better than the 6-species-3-reactions Fluent mechanism. In other words, as the heat transfer depends on both the flame temperature and species, the heat transfer will be quantitatively well predicted once the flame combustion is modeled accurately. Therefore, a conclusion can be made that for quantitatively analysis of both flame and heat transfer, using a detailed mechanism is better than a reduced mechanism, which in turn is better than using an isothermal hot flow. However, it is seen from this study that the simulation cost by the Chemkin mechanism is more than doubled compared to the Fluent mechanism, and there are cases where only the heat transfer modeling is concerned by engineers, where only a qualitative analysis is desired, especially for multiple flames simulation. In such case, a reduced mechanism is preferred due to its lower computation cost than a detailed one. There are even some studies proposing that isothermal jet and flame jet are comparable in terms of impingement heat transfer [35,36]. Obviously, the hot flow simulation misses flame reaction zone and its usage is limited to much high nozzle-plate distance. Therefore, the Fluent mechanism is used to further check measured heat flux. The consequence is that due to the heated lower surface, the temperature difference between the lower and upper surfaces of the plate is lower, resulting in lower heat flux measured. So, the experimental heat flux is lower than the computed heat flux because in the model the plate temperature is fixed at the wall boundary.
whether the variation trend of heat flux can be correctly predicted or not at various operational conditions.

The radial heat flux profiles at lower nozzle-plate distances of $H = 30, 18$ and $6$ mm for the flame operating at $Re = 1500, \Phi = 1.0$ are shown in Fig. 8. The experimental measured heat flux profiles are denoted by dashed lines, while the computed profiles by solid lines. It is observed that at each value of $H$, the computed heat fluxes are lower than the measured ones in the central region due to the underestimate of the numerical model. While at the further radial position, the computed heat fluxes are not definitely higher than the experimental data. This is because with decreasing nozzle-plate distance, the flame covers a larger area of the wall surface so that the heat fluxes in the outer radial position are close to each other between the experimental and numerical data.

Fig. 8 also indicates that the radial positions of the main and sometimes second peaks of the measured heat flux are followed quite well by the simulation. The second peak occurs at small nozzle-plate distance and is well observed in both isothermal jet and flame jet impingement heat transfer [37,38]. Here, a summary can be made that though the model needs further improvement to solve the quantitative underestimate problem, the radial heat flux distribution can be qualitatively well predicted by the present model. A similar study conducted by Tajik et al. [22] also points out that a large discrepancy of 25% exists between modeling result and experimental data.

3.4.2. Effect of $\Phi$

The effect of equivalence ratio on heat transfer is examined at fixed $Re = 1500$ and $H = 42$ mm. As the equivalence ratio deviates from stoichiometric $\Phi = 1.0$, the flame temperature decreases due to incomplete combustion, and the flame speed is also reduced leading to a longer flame cone height. The cone height is the longest at $\Phi = 0.8$, followed by $\Phi = 1.2$ and is the shortest at $\Phi = 1.0$, consistent with the trend in Ref. [39].

As shown in Fig. 9, the heat flux at the stagnation point is the highest at $\Phi = 1.2$ because in this case the flame tip is the closest to the wall, and its high temperature contributes to the local heat transfer. The cone height of the flame at $\Phi = 0.8$ is over $42$ mm, and thus the cone is truncated by the target surface, such that the stagnation point heat flux is nearly zero due to the contact of unburned reactant mixture with the surface. Furthermore, as a result of lowered flame temperature at either $\Phi = 0.8$ or $\Phi = 1.2$, the heat fluxes beyond $r = 5$ mm becomes lower than those at $\Phi = 1.0$. However, at $r > 60$ mm, the heat fluxes at $\Phi = 1.2$ are the highest among the three cases. The reason is simply that the total fuel flow as well as the thermal energy input is increased with increasing equivalence ratio, and there is more fuel to be burned in the far radial position for the flame at $\Phi = 1.2$. The findings of this session are consistent with the effect of equivalence ratio on the experimental heat flux distribution [40].

3.4.3. Effect of $Re$

Fig. 10 examines the effect of Reynolds number on the heat transfer of the flame at $\Phi = 1.0$ and $H = 42$ mm. Compared to the flame at $Re = 1500$, a higher Reynolds number of 2000 increases the flame cone height beyond $42$ mm due to the higher jet velocity. Thus, it is the potential core that is in contact with the impingement plate at the stagnation point. The potential core consists of unburned cold reactants, and thus the heat transfer from it to the plate is low, resulting a dent in the radial heat flux profile at $Re = 2000$. However, as a higher fuel flow rate is supplied at $Re = 2000$ in comparison to $Re = 1500$, there is more fuel for burning beyond the flame cone. Hence, the thermal boundary layer would thicken and extend radially in the wall jet region, leading to higher heat fluxes at $r > 10$ mm. Not that the results here agree well with the effect of Reynolds number on the experimental heat flux distribution [40].

4. Conclusion

A numerical model was constructed in this study, the objective of which is to evaluate the model performance of predicting flame impingement heat transfer. A round Bunsen flame impinging onto a copper plate was modeled by Fluent software and mesh independence check was conducted by using the 2-step reaction mechanism coherent in the software. Next, both the flame and heat transfer characteristics of the impinging flame predicted by the Fluent mechanism and a Chemkin-compatible mechanism were computed and compared. The main findings are as follow.

Both mechanisms over predicted the height and maximum temperature of the impinging flame. Temperatures are the highest above the flame tip in the Fluent mechanism case and off the tip in the Chemkin mechanism case. These hot spots influence the impingement heat transfer under certain conditions. Overall, the Chemkin mechanism gives a more reasonable temperature field and better predicts flame height and flame structure due to the more species and reactions considered by the mechanism.

The heat transfer results showed there are other possibilities of model improvement, i.e. modeling the plate thickness. As far as the two mechanisms are concerned, although the heat fluxes in the stagnation and near wall jet regions are underestimated, both mechanisms have the capability to follow the experimental heat flux variation. Overall, the Chemkin mechanism presents better consistence with experiments due to the better predicted flame temperature and structure, but the running time may be too long for engineers. Moreover, such detailed information on flame structure may be out of interest of engineers. So, when only heat transfer is desired, the Fluent mechanism is a good choice for providing a qualitative analysis, with acceptable low computation cost.

Acknowledgement

The authors thank the National Natural Science Foundation of China (51766003) for financial support of the present study.

References


