Numerical Simulations of Flat Laminar Premixed Methane-Air Flames at Elevated Pressure

M. Goswami\textsuperscript{a}, K. Coumans\textsuperscript{a}, R. J. M. Bastiaans\textsuperscript{a}, A. A. Konnov\textsuperscript{b} & L. P. H. de Goey\textsuperscript{a}

\textsuperscript{a} Combustion Technology Section, Mechanical Engineering, Eindhoven University of Technology, Eindhoven, The Netherlands
\textsuperscript{b} Division of Combustion Physics, Lund University, Lund, Sweden

Published online: 30 Sep 2014.

To cite this article: M. Goswami, K. Coumans, R. J. M. Bastiaans, A. A. Konnov & L. P. H. de Goey (2014) Numerical Simulations of Flat Laminar Premixed Methane-Air Flames at Elevated Pressure, Combustion Science and Technology, 186:10-11, 1447-1459, DOI: 10.1080/00102202.2014.934619

To link to this article: http://dx.doi.org/10.1080/00102202.2014.934619

PLEASE SCROLL DOWN FOR ARTICLE
Numerical Simulations of Flat Laminar Premixed Methane-Air Flames at Elevated Pressure

M. Goswami,1 K. Coumans,1 R. J. M. Bastiaans,1 A. A. Konnov,2 and L. P. H. de Goey1

1Combustion Technology Section, Mechanical Engineering, Eindhoven University of Technology, Eindhoven, The Netherlands
2Division of Combustion Physics, Lund University, Lund, Sweden

Two-dimensional axisymmetric simulation of stoichiometric methane-air flames stabilized on flat burners at elevated pressure is reported in the present work. Such flames, in practice, are experimentally obtained using the heat flux method for measurement of laminar burning velocity of fuel-oxidizer mixtures (Bosschaart and de Goey, 2004; Goswami et al., 2013). The method makes use of a burner with a perforated brass burner plate. The dimensions of such a plate play an important role in creating flat flames. The present investigation is focused on studying laminar premixed flame structure numerically at elevated pressure up to 15 bar using a one-step and a detailed chemical reaction mechanism. Three burner plate models (of varying hole diameter and porosity) are used in the simulations for pressures up to 7 bar with a one-step mechanism. The surface area increase of the flame was evaluated based on an isotherm at 900 K and the net reaction rate of methane compared to a flat flame. The comparison of these models shows that the surface area increase can significantly be reduced by choosing a smaller hole diameter and larger porosity. The results of the detailed simulations using an appropriate chemical reaction mechanism up to 15 bar using a burner plate model, which is similar to the ones used in experiments (mentioned above), show a nonlinear increase of the flame curvature with elevating pressure. A hole diameter of 0.25 mm and a pitch of 0.29 mm is suggested for a burner plate in such experiments. Flame structure at elevated pressure is also analyzed further based on species profiles obtained.

Keywords: Elevated pressure; Heat flux method; Laminar burning velocity; Methane combustion

Introduction

Burner stabilized flames are used in many domestic and industrial applications. In recent literature, flat flame burners are used for many experimental purposes in fundamental research, for example, determining fuel characteristics like laminar burning velocity (Bosschaart and de Goey, 2004), studying flame structure (Ratna Kishore et al., 2011), and flame stabilization and blowoff studies (Kedia and Ghoniem, 2011). Many applications
like gas turbines operate at elevated pressures and temperatures. Therefore, experimental and numerical flame studies at elevated pressures is an active area of research in the field of combustion.

The laminar burning velocity defines the rate at which the unburned mixture is consumed in a propagating laminar flame. This parameter is considered one of the most important properties in assessing flame stability and flashback in practical systems like burners and combustors. Apart from its importance in designing combustors, this parameter also holds key responsibility in validating chemical reaction mechanisms. The heat flux method is an experimental technique employed to determine unstretched laminar burning velocity. This method employs a burner, which stabilizes a flat flame under near adiabatic conditions. The burner houses a perforated plate through which unburned gas is guided out and burned. The main feature of this method is that the heat loss from the flame is compensated by adding heat to the unburned gas mixture. A perforated plate is fitted on a burner head, which is maintained at a temperature higher than the unburned gas temperature. This gives a heat transport from the head to the burner plate and finally to the unburned gas mixture. A detailed study of Bosschaart and de Goey (2003) describes the principle, working, and errors involved in the heat flux method. Burning velocity results of a variety of fuels can be found in the literature using this method at various conditions of pressure, temperature, and fuel-oxidizer composition. The extension of this method to elevated pressure was performed recently by Goswami et al. (2013) using methane-air flames up to 5 atm.

The temperature profiles obtained from the perforated burner plate in the heat flux method determine whether a flame is adiabatic or not (Bosschaart and de Goey, 2003). The burner plate creates a pressure drop in the flow and damps out any fluctuation downstream. Therefore, the plate plays an important role in stabilizing a flat flame. The specifications of the burner plate are given by the plate diameter, the hole diameter, plate thickness, and the pitch. The pitch ($s'$) is given by the distance between the center of two consecutive holes. Another quantity, porosity ($\chi$) is defined as the ratio of area covered by the holes to the total area. It signifies the voids within the plate. The numerical simulations performed by de Goey et al. (1995) at atmospheric pressure showed that methane-air flame experienced small curvature with inlet velocity higher than 40 cm/s with a burner plate having hole diameter ($d$) = 0.5 mm and pitch ($s'$) = 0.7 mm. Other simulations (Somers, 1994) showed that a decrease in hole diameter can reduce the flame curvature. It was also shown that very small porosity ($\chi$) will lead to early blow off due to high velocity in the holes. The relationship between porosity ($\chi$) and pitch ($s'$) is described in the subsequent section.

Under conditions of elevated pressure, if the upstream flow fluctuations are not damped in time, the flame may show curvature due to distortion in heat and mass transport as shown in Figure 1 that depicts the cross section of a burner plate. The rectangular part represents the plate with holes between them. These curvature effects may not be visible to the naked eye. The increase in surface area may, hence, influence the burning velocity.

Figure 1 Flame curvature above the burner plate.
Table 1  Dimensions of the burner plate models

<table>
<thead>
<tr>
<th>Model</th>
<th>Hole diameter (mm)</th>
<th>Porosity $\chi$ (−)</th>
<th>Pitch $s'$ (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.5</td>
<td>0.63</td>
<td>0.6</td>
</tr>
<tr>
<td>2</td>
<td>0.3</td>
<td>0.63</td>
<td>0.36</td>
</tr>
<tr>
<td>3</td>
<td>0.3</td>
<td>0.51</td>
<td>0.4</td>
</tr>
</tbody>
</table>

measured at elevated pressure. Therefore, the objective of the present study is to focus on the question if the flatness of stoichiometric methane (CH$_4$) - air flames obtained from different burner plate designs or models depends on the plate hole diameter and distance between successive holes. In order to assess how a flat (heat flux method) flame behaves in a high pressure environment, 2D axisymmetric simulations have been performed on three burner plate models. The first model is with $d = 0.5$ mm and $\chi = 0.63$. The second burner plate model has a reduced diameter of $d = 0.3$ mm but similar porosity $\chi = 0.63$. Finally, the third burner plate model, which is also used in high pressure experiments (Goswami et al., 2013), is with $d = 0.3$ mm and reduced porosity $\chi = 0.51$. The three models are summarized in Table 1.

AXISYMMETRIC SIMULATIONS

The 2D axisymmetric simulations are performed for a stoichiometric methane-air mixture with an unburned gas temperature of 300 K using a commercially available CFD code Ansys Fluent (2011). Three different burner plates, as summarized in Table 1, are modeled. Steady state simulations with an incompressible flow and a one-step kinetic mechanism are performed up to 7 bar. Earlier studies (de Goey et al., 1995) present a comparison of a one-step mechanism with the skeletal mechanism, introduced by Smooke (1991). The results showed only small differences in the calculated temperature profile, which is used to determine the flame curvature. Therefore, it was concluded that the one-step mechanism is well suited to predict the flame shape and curvature effects (de Goey et al., 1995). Detailed simulations using the DRM19 mechanism (Kazakov and Frenklach, 1995) are also performed up to 15 bar, with model 3 that resembles the burner plate from the experimental setup.

Governing Equations

The combustion of gases is governed by a set of equations describing the conservation of mass, momentum, energy, and species. The equations are given by:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \quad (1)$$

$$\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\vec{\tau}) + \rho \vec{g} + \vec{F} \quad (2)$$

$$\nabla \cdot (\vec{v} (\rho E + p)) = \nabla \cdot \left( k \nabla T - \sum_i h_i \vec{J}_i + (\vec{\tau} \cdot \vec{v}) \right) + S_h \quad (3)$$
where \( v \) is the gas mixture velocity and \( \rho \) the density, \( p \) is the static pressure, \( \tau \) is the stress tensor, and \( \vec{g} \) and \( \vec{F} \) are the gravitational body force and external body forces. \( k \) is the thermal conductivity and \( \vec{J}_i \) is the diffusion flux of species. The first three terms on the right-hand side of Eq. (3) represent energy transfer due to conduction, species diffusion, and viscous dissipation, respectively. \( S_h \) is the source term, which includes the heat of chemical reaction for reactive flows. The total specific energy \( E = h - \frac{p}{\rho} + \frac{v^2}{2} \), where sensible enthalpy \( h \) is defined for ideal gases as \( \sum_i Y_i h_i \), with \( Y_i \) the mass fraction of species \( i \) and \( h_i = \int_{T_{ref}}^T c_{p,i} \, dT \), where \( T_{ref} \) is 298.15 K. The source term \( S_h \) is essentially the energy produced due to chemical reaction, which is given by \( S_h = -\sum_i h_i^0 R_i \), where \( h_i^0 \) is the enthalpy of formation of species and \( R_i \) is the volumetric rate of creation of species \( i \).

For solving conservation equations for chemical species, the solver predicts the local mass fraction of each species, \( Y_i \), through the solution of a convection-diffusion equation for the \( i \)th species. This species conservation equation has the general form:

\[
\frac{\partial}{\partial t} (\rho Y_i) + \nabla \cdot (\rho \vec{v} Y_i) = -\nabla \cdot \vec{J}_i + R_i
\]

where \( R_i \) is the net rate of production of species \( i \) by chemical reaction. Equation (4) needs to be solved for \( N_s - 1 \) species (where \( N_s \) is the total number of species), since the mass fraction of one of the species can be solved using the constraint that the sum of all the species mass fractions equals 1. To minimize the numerical error, the species that is present in abundance is chosen for this purpose. In the case of methane-air mixture it is nitrogen. The chemical source terms are computed in Fluent by the laminar finite-rate model using Arrhenius expressions from the reaction mechanism file. The net source of chemical species due to reaction \( R_i \) is computed as the sum of the Arrhenius reaction sources over the reactions that the species participate in:

\[
R_i = M_{w,i} \sum_{r=1}^{N_s} \hat{R}_{i,r}
\]

where \( M_{w,i} \) is the molecular weight of species \( i \) and \( \hat{R}_{i,r} \) is the Arrhenius molar rate of creation or destruction of species \( i \) in reaction \( r \).

The stiff laminar flames are modeled with the laminar finite-rate model using the pressure-based solver. For steady simulations, this option approximates the reaction rate \( R_i \) in the species transport equation (4) as:

\[
R_i^* = \frac{1}{\tau} \int_0^\tau R_i \, dt
\]

where \( \tau \) is a time step, with the default value set to one-tenth of the minimum convective or diffusive time-scale in the cell.

The solver (Ansys Fluent, 2011) uses a control-volume-based technique to convert the governing equations to algebraic equations that can be solved numerically. The second-order upwind scheme is used for spatial discretization of all the governing equations. The pressure-based coupled algorithm is used to enable full pressure-velocity coupling. This algorithm solves the momentum and pressure-based continuity equations together, which improves the rate of solution convergence when compared to the segregated algorithm.
The standard pressure interpolation scheme is used to interpolate the pressure values at the faces. The double-precision solver is utilized for solving the discretized set of algebraic equations. To obtain a converged solution, a two-step solution process was adopted. First, the flow, energy, and species equations were solved with the reactions disabled. This cold flow solution was obtained within 1000 iterations and provided a good starting point for the combustion problem. With the basic flow pattern now established, the second step was to enable the reactions and ignite the mixture. The solution was converged to residuals of the species, continuity, and energy equations of less than $10^{-5}$, $10^{-4}$, and $10^{-6}$, respectively. The solution was then ran until the continuity equation residual dropped below $5 \times 10^{-5}$, to ensure that the solution does not change anymore, so that a steady-state is reached.

The density, $\rho$, of the multicomponent mixture is calculated using the ideal gas law for an incompressible flow. The viscosity, $\mu$, and the thermal conductivity, $k$, for the individual species are computed using the kinetic theory. These two properties of the mixture are calculated with the ideal gas mixing law, which is allowed when the ideal gas law is used for the density. The inputs to the calculation of viscosity, thermal conductivity, specific heat, and mass diffusion coefficient of individual species are taken from the chemical reaction mechanism, which includes thermodynamic and transport properties of the species. The Soret effect is taken into consideration in computing the diffusion flux.

### Computational Domain

The three burner plate models investigated in this work are described in Table 1. The burner plate used in the experiments is a round perforated plate with a diameter of 30 mm and a thickness of 1 mm (model 3). The holes have a diameter of $d = 0.3$ mm and a pitch of $s' = 0.4$ mm. The holes are placed in a hexagonal pattern in order to stabilize a flat flame above the burner plate. The actual plate has over 5000 holes, but a schematic representation is shown in Figure 2. The presence of symmetry planes can be used to isolate a small hexagonal unit cell; this unit cell is approximated by an axisymmetrical cell. The red rectangle represents the plane that needs to be modeled, with the axis of rotation through the center of a hole. The round edge of the cell is modeled as a symmetry boundary. The right side of Figure 2 shows that when the actual pitch, $s'$, is used in the model, there will be parts of the burner plate that are not taken into account. This will lead to a different porosity than the actual plate. To compensate for this, the pitch, $s$, is now chosen such that the porosity of the axisymmetrical cell (axi) is equal to the porosity of the hexagonal unit cell (hex):

![Figure 2 Burner plate with hexagonal pattern for holes. Red plane is the computational domain, which is rotated along the axis through the center of the hole.](image-url)
The axisymmetric axis is defined along the \( x \)-axis and goes through the center of a hole. The round edge of the domain is modeled as a symmetry boundary and the burner plate as a wall. The interaction between the neighboring holes in the actual burner plate geometry will differ from the axisymmetric model, which may introduce a small error. The present study, however, focuses on the trends of the observed surface area increase in a qualitative view. The in-flow of the premixed gases is defined as a velocity inlet, where the velocity magnitude and the species mass fractions are defined. The blue line in Figure 3 (top) represents the approximate position of the flat flame. The exhaust gases leave through the pressure outlet, which is placed sufficiently far from the flame. The length of the computational domain is determined by the length of the in-flow area \( L_{in} \), the thickness of the burner plate \( L_{hole} \), and the length of the flame/outflow area \( L_{out} \). Typical dimensions are 1, 1, and 5 mm, respectively. The computational grids were made in Gambit 2.4.6, which is Fluent’s preprocessor for creating the geometry and the mesh. The grid is shown in Figure 3 (bottom). The grid consists of several sections, labeled A to E, with an increasing level of refinement. Initially, section E has the same cell dimensions as section D. For several cases, the whole grid was refined in Fluent after the solution was converged. This was done by adaptation of the grid, which creates four cells out of one. The solution was then converged again to ensure that it remains the same. This shows that the used cell dimensions provide a grid independent solution. It was found that the chosen cell dimensions for the grids used in this work are small enough. For all the simulations, the grid was refined in the flame region from D to E, after the solution was converged. The largest gradients occur in this region, so the refinement was used to check that the solution remains grid independent for all cases.

\[
\chi_{axi} = \left( \frac{d}{s} \right)^2
\]

\[
\chi_{hex} = \left( \frac{\pi}{2\sqrt{3}} \right) \left( \frac{d}{s} \right)^2
\]
Chemical Reaction Mechanism

In order to perform the detailed simulations, an appropriate chemical reaction mechanism has to be selected. One-dimensional simulations were performed with stoichiometric methane-air flame using four different chemical reaction mechanisms. The pressure was increased from 1 to 30 bar. An overview of the calculated adiabatic burning velocities, together with the experimental results, is presented in Figure 4. All four mechanisms show a trend where the burning velocity is decreasing nonlinearly with increasing pressure.

The GRImech 3.0 (Smith et al., 1999) has 53 species in the scheme and shows the best agreement with the experimental results. Only at a pressure of 4.5 bar does the measured burning velocity deviate approximately by 1 cm/s. The mechanism by Konnov (2009) is accurate at atmospheric pressure and shows a large deviation from GRImech 3.0 for elevated pressures. The DRM19 and DRM22 mechanisms (Kazakov and Frenklach, 1995) (consisting of 19 and 22 species, respectively) both show higher velocities at atmospheric pressure (39.3 and 39.6 cm/s, respectively). The DRM19 mechanism shows deviations between 7% and 11% compared to the GRImech 3.0 mechanism for pressures ranging from 1 to 15 bar. For higher pressures the predictions show a better agreement. The velocities calculated with the DRM22 mechanism are slightly higher than those of the DRM19 mechanism over the entire pressure range and therefore deviate more from the GRImech 3.0. Although the experimental results show the best agreement with GRImech 3.0, the DRM19 mechanism is selected due to lesser number of species for the detailed simulations, which ensures faster computations. Before simulating the flame using the detailed DRM19 mechanism, the standard one-step mechanism was also implemented for reasons discussed before. The 2D simulations in the present work predicted the same values of burning velocity as 1D simulations up to a pressure of 12 bar. A validation of the 2D simulations was performed by comparing the experimental results with the predicted
RESULTS AND DISCUSSION

To determine the curvature and surface area increase of the flame, an indicator for the flame stand-off distance is introduced. The parameter $\delta T_{iso}$ gives the distance from the burner plate outlet to a certain isotherm with value $T_{iso}$, and varies from the center boundary to the side boundary, as indicated in Figure 5. $\delta_{900}$ is the parameter used for $T_{iso} = 900$ K. The center boundary is the axisymmetric axis that is defined through the center of the hole and the side boundary is the symmetry boundary condition that is located above the burner plate. In previous studies (de Goey et al., 1995), the 900 K isotherm has been chosen to determine the curvature, as this is the approximate temperature above which chemical reactions become important. The isotherms for the pressures 1, 5, and 10 bar are illustrated in Figure 5. The corresponding laminar burning velocities of the three adiabatic cases are used as the inlet velocity. This shows that the flame is flat, when the stand-off distance of the 900 K isotherm is equal along the two boundaries. This is the case at 1 bar and the flame exhibits curvature as the pressure increases. The integration of the 900 K isotherm from center to side boundary and then surface revolution along the center boundary gives the surface area of the curved flame. An increase of the surface area ($S_{incr}$) is then obtained by comparing it to an ideal flat flame (a circle of diameter 30 mm). The surface area increase, according to the 900 K isotherm, is calculated for all the simulations performed with the one-step mechanism and the DRM19 mechanism.

Instead of using the 900 K isotherm to determine the flame curvature, it is also possible to define a different parameter. One of the post-processing options is to present the net reaction rate of methane, $R_{CH_4}$. This is shown for the simulations at 10 bar in Figure 6. The curvature of the layer where the reactions take place can also be regarded as a suitable indicator for the flame curvature. The net reaction rate is plotted along the center and the side boundary (Figure 6). The shift that occurs between the two plots, $\delta_{CH_4}$, is essentially the

![Figure 5](https://example.com/figure5.png)

**Figure 5** $\delta T_{iso}$ for 1, 5, and 10 bar (top to bottom) of stoichiometric methane-air flame and 300 K (unburned gas temperature).
same principle as the parameter $\delta_{900}$, used to determine the curvature of the 900 K isotherm. When the flame is entirely flat, the plots are exactly the same and on top of each other. The shift can be measured by determining the curve length between the peaks of the plots.

Figure 6  Net reaction rate of stoichiometric methane-air flame at 10 bar and 300 K (unburned gas temperature).

Figure 7  Surface area increase of the stoichiometric methane-air flame for the three burner plate models. The inlet velocities mentioned are the experimentally determined laminar burning velocity of stoichiometric methane-air flames at corresponding pressures.
Simulations Using One-Step Mechanism

The surface area increase, $S_{incr}$, has been calculated according to both the 900 K isotherm and the net reaction rate of methane. The results, using the one-step mechanism and 900 K isotherm ranging from $p = 1$ to 7 bar, are presented in Figure 7. The experimentally determined burning velocities (Goswami et al., 2013) were used as the inlet velocity in the models. The results suggest a nonlinear increase of $S_{incr}$ with elevating pressure. The results in Figure 7 show that the amount of surface area increase strongly depends on the geometry of the burner plate. The amount of $S_{incr}$ is reduced by a factor of 3 when burner plate (2) (black triangles) with $d = 0.3$ mm, $\chi = 0.63$ is used when compared to burner plate (1) (red diamonds) with $d = 0.5$ mm, $\chi = 0.63$. This reduction is achieved by decreasing the hole diameter from 0.5 to 0.3 mm, while the porosity remains constant. The results from burner plate (3) (blue circles) with $d = 0.3$ mm, $\chi = 0.51$, are compared to burner plate (2) with $d = 0.3$ mm, $\chi = 0.63$ to find the influence of the porosity. The amount of $S_{incr}$ is reduced by a factor of 2 when the porosity is increased from $\chi = 0.51$ to $\chi = 0.63$. The significant increase in porosity is achieved by decreasing the pitch of the burner plate by only 0.04 mm.

Detailed Simulations

The detailed simulations using DRM19 mechanism were performed up to 15 bar and show the surface area increase more clearly in Figure 8. For 1 and 5 bar, the results from the one-step mechanism are included. This shows the influence of using the detailed reaction mechanism. The difference is small between the DRM19 mechanism and the one-step mechanism. At 5 bar, the surface area increases are 1.7% and 2.2%, respectively. The simulations at 1 bar show a negligible increase for both cases. This suggests that the one-step mechanism is sufficient to determine the approximate surface area increase. The results at 10 and 15 bar clearly show a nonlinear increase of the $S_{incr}$ with elevating pressure. The surface area increase determined from the maximum reaction rate of CH$_4$ shows the same nonlinear increase. However, the values are smaller compared to the 900 K isotherm criterion. At 5 bar the $S_{incr}$ is only 0.45% but it rises quickly to 3.4% and 11.8% for 10 and 15 bar, respectively.

The detailed species structure of the flat flame is presented in Figure 9. The profiles of the major species mass fraction along the center and side boundary are compared for the cases at 1 and 10 bar. This shows the influence of the flame curvature and increase in pressure. The dotted vertical blue line represents the location of the burner plate. The case at 10 bar shows the largest difference between both boundaries. This is caused by the large curvature that was already shown in Figures 5 and 6, for the temperature isotherms and the reaction rate of methane. The profiles along the side boundary are shifted towards the burner plate compared to the center boundary. They coincide after the reaction layer, where the flow becomes uniform again and the mixture reaches chemical equilibrium.

The influence of the pressure becomes clear when comparing the two cases of 1 and 10 bar. At elevated pressure, the collision rates of molecules increase, which cause the reactions to take place at a higher rate. This decreases the thickness of the reaction layer, which is shown by the steeper mass fraction profiles and the mixture becomes uniform very fast. The maximum mass fractions of H, H$_2$, and CO are approximately two times lower at 10 bar, which is a result of the high reaction rate. Simulations showed that the maximum reaction rate of methane is approximately 15 times higher at 10 bar compared to atmospheric pressure. Highly mobile hydrogen radicals promote the burning rate of the mixture. It is possible that these radicals recombine at the burner surface, which will influence the
measured burning velocity. It can be determined whether there is an increase of this effect with pressure. The main sources of H radicals during the combustion process are the CO oxidation reaction 9 and the HCO decomposition reaction 10 (Hu et al., 2009).

\[
\text{OH} + \text{CO} \leftrightarrow \text{H} + \text{CO}_2 \tag{9}
\]

\[
\text{HCO} + \text{H}_2\text{O} \leftrightarrow \text{H} + \text{CO} + \text{H}_2\text{O} \tag{10}
\]

The potential influence of burner surface reactions may take place as the radicals might reach the surface. An experimental study was performed by Hermanns (2007) at ambient conditions by using a gold-coated burner plate and a conventional brass burner plate. It was observed that the laminar burning velocity was not influenced. In the present work, the species profiles show an increase of H$_2$ on the burner plate, compared to the center boundary. The H radicals are almost zero at the burner plate and show a peak in the reaction layer. For the cases at 1 and 10 bar, the concentration of H radicals at the burner plate are six orders of magnitude smaller compared to the maximum present in the flame. This shows that the increase in pressure does not influence surface reactions at the burner plate.

To minimize the flame curvature effects at elevated pressure, the hole diameter must be reduced and the porosity must be increased. This means that one has to stay within a certain pressure for given geometries or optimize the geometry further. Presently, the holes are drilled in the brass plate and a hole diameter of $d = 0.25$ mm should be considered as the lower limit. Somers (1994) recommended to use porosities higher than $\chi = 0.67$. It was found that the porosity of $\chi = 0.8$ can be considered as an upper limit to avoid corrosion of the burner plate material. The burner plate that is expected to show the least amount of

![Figure 8](Image)

**Figure 8** Surface area increase of the flame for the burner plate model 3, which resembles the plate used in experiments. The DRM19 mechanism is used for both the criteria, namely, 900 K and maximum reaction rate of methane.
Figure 9 Species profiles at (left) 1 bar and (right) 10 bar of stoichiometric methane-air flames using DRM19 mechanism for burner plate model 3 \((d=0.3\,\text{mm}, \chi=0.51)\). Temperature (T) profile is also indicated. The dotted vertical blue line represents the location of the burner plate.

The surface area increase should, therefore, have a hole diameter of \(d=0.25\,\text{mm}\) and a pitch of \(s=0.29\,\text{mm}\). This corresponds to a porosity of \(\chi=0.67\).

CONCLUSIONS

Two-dimensional axisymmetric simulations were performed on stoichiometric methane-air flat flames using three burner plate models used in the heat flux method at elevated pressures. The surface area increase of the curved flames, compared to a perfectly flat flame is calculated based on an isotherm at 900 K and the net reaction rate of methane. Both show the same trend in surface area increase with elevating pressure. Three burner plates were geometrically modeled for pressures up to 7 bar with a one-step mechanism. The comparison of these models shows that the surface area increase can significantly be reduced by choosing a smaller hole diameter and larger porosity. The results of the detailed simulations using a DRM19 chemical reaction mechanism up to 15 bar show a nonlinear increase of the flame curvature with elevating pressure. The surface area increase of the methane-air flame is around 16% at 15 bar with the burner plate that is used in the heat flux method experimental setup. This must be reduced by manufacturing a new burner plate with a smaller hole diameter and a higher porosity. A hole diameter of 0.25 mm and a pitch of 0.29 mm is suggested for a burner plate in such experiments.

FUNDING

Financial support of this work by the European Union within the ‘H2-IGCC’ Project (Grant Agreement No.: 239349) is gratefully acknowledged.
REFERENCES


