Reynolds number and geometry effects in laminar axisymmetric isothermal counterflows

Gianfranco Scribano and Fabrizio Bisetti

Citation: [Phys. Fluids](#) **28**, 123605 (2016); doi: 10.1063/1.4972238

View online: <http://dx.doi.org/10.1063/1.4972238>

View Table of Contents: <http://aip.scitation.org/toc/phf/28/12>

Published by the [American Institute of Physics](#)

Articles you may be interested in

[On the flow structures and hysteresis of laminar swirling jets](#)

[Phys. Fluids](#) **28**, 123604 (2016); 10.1063/1.4972227

[Finding order in complexity: A study of the fluid dynamics in a three-dimensional branching network](#)

[Phys. Fluids](#) **28**, 123602 (2016); 10.1063/1.4971315

Reynolds number and geometry effects in laminar axisymmetric isothermal counterflows

Gianfranco Scribano¹ and Fabrizio Bisetti^{2,3,a)}

¹*Department of Mechanical, Materials and Manufacturing Engineering, University of Nottingham Malaysia Campus, Jalan Broga, Semenyih 43500, Selangor Darul Ehsan, Malaysia*

²*Department of Aerospace Engineering and Engineering Mechanics, University of Texas at Austin, Austin, Texas 78712-1085, USA*

³*Clean Combustion Research Center, King Abdullah University of Science and Technology, Thuwal 23955, Saudi Arabia*

(Received 1 May 2016; accepted 2 December 2016; published online 29 December 2016)

The counterflow configuration is a canonical stagnation flow, featuring two opposed impinging round jets and a mixing layer across the stagnation plane. Although counterflows are used extensively in the study of reactive mixtures and other applications where mixing of two streams is required, quantitative data on the scaling properties of the flow field are lacking. The aim of this work is to characterize the velocity and mixing fields in isothermal counterflows over a wide range of conditions. The study features both experimental data from particle image velocimetry and results from detailed axisymmetric simulations. The scaling laws for the nondimensional velocity and mixture fraction are obtained as a function of an appropriate Reynolds number and the ratio of the separation distance of the nozzles to their diameter. In the range of flow configurations investigated, the nondimensional fields are found to depend primarily on the separation ratio and, to a lesser extent, the Reynolds number. The marked dependence of the velocity field with respect to the separation ratio is linked to a high pressure region at the stagnation point. On the other hand, Reynolds number effects highlight the role played by the wall boundary layer on the interior of the nozzles, which becomes less important as the separation ratio decreases. The normalized strain rate and scalar dissipation rate at the stagnation plane are found to attain limiting values only for high values of the Reynolds number. These asymptotic values depend markedly on the separation ratio and differ significantly from the values produced by analytical models. The scaling of the mixing field does not show a limiting behavior as the separation ratio decreases to the smallest practical value considered. *Published by AIP Publishing.* [<http://dx.doi.org/10.1063/1.4972238>]

I. INTRODUCTION

Broadly defined, a counterflow setup consists of two steady opposed jets, usually axisymmetric, impinging on each other and establishing a stagnation flow.^{1–3} If the streams issuing from the two nozzles differ in composition, a mixing layer forms across the stagnation plane. A schematic representation of the flow configuration investigated in this paper is given in Figure 1, where the two impinging jets issue from contoured nozzles (see Figure 1(a)).

Ease of optical access away from solid surfaces, stability of the flow, and selective control over the strain rate and rates of mixing make the steady counterflow setup a widely employed configuration to study the interaction between heat and mass transport and other molecular processes of interest, e.g., chemical reactions. For example, in nonpremixed counterflow flames, reactions and heat release occur inside the thin mixing layer across the stagnation plane.^{1,4} The structure⁵ and

^{a)}Electronic mail: fbisetti@utexas.edu

FIG. 1. Schematic of the axisymmetric flow configuration and cylindrical coordinate system (fluid moves downward). z and r are the axial and radial coordinates, respectively. (a) Overview of the flow configuration, featuring two nozzles of diameter D separated by a distance H and centered with respect to the stagnation plane. (b) Detail of the region of interest and contoured wall near the stagnation plane. The nozzle radius is $R = D/2$, $L = H/2$ is half the separation distance between the two nozzles, and the bulk velocity U , computed at the nozzle exit plane (area is $\pi D^2/4$) is the same for both nozzles. δ_w and δ_z indicate the thickness of the wall boundary layer and that of the mixing layer across the stagnation plane, respectively.

aerodynamic extinction^{6,7} of nonpremixed flames are often studied in counterflow burners for varying strain rates.⁸ Recently, the nucleation and growth of condensing aerosols have been investigated numerically in a counterflow by one of the authors.⁹

In the range of operating conditions relevant to laminar flow applications, the flow field is inviscid and irrotational over most of the domain (see Figure 1(b)). Vorticity and viscous effects are confined to the wall boundary layer on the interior of the nozzles and along the external shear layer that extends from the nozzle wall outwards in the radial direction. The region around the stagnation point conforms closely to the potential stagnation flow model,¹⁰ whereby the velocity field varies linearly with the axial and radial coordinates and is described by a single parameter, the strain rate at the stagnation point. Away from the stagnation point, the velocity field is more complex and affected by the geometry and operating conditions of the counterflow. At large enough Reynolds numbers, mixing between the two streams occurs inside the potential flow region.

The details of the velocity and mixture fraction fields in counterflows have been investigated in the past, with most studies related to Ref. 10 or directly involving nonpremixed flames.^{11,12} One finds that nonpremixed flames are sensitive to the mixing field, which in turn is affected by the flow rates through the nozzles and the nozzles' geometry. A quantitative characterization of the velocity and mixing fields is recognized as an important prerequisite for the analysis of flames in counterflow burners.⁸ Recently, there has been renewed interest in studying the details of the flow field in counterflow burners^{7,13–15} with the aim of reconciling the differences between the actual flow field and that assumed by reduced order models,^{16–18} which are adopted to describe counterflow flames numerically and analytically.

For the purpose of the analysis presented in this work, the parameters characterizing the flow are the distance between nozzles' exits (H), the diameter of the exits (D), the bulk velocity at the exit sections ($U = Q/A$, where $A = \pi D^2/4$ is the exit area and Q is the volumetric flow rate, taken to be equal in each nozzle), and the kinematic viscosity of the fluid (ν), which is taken to be constant. The straightforward application of dimensional analysis to the counterflow configuration in Figure 1 identifies $Re_L = UL/\nu$ and $\alpha = H/D$ as two nondimensional groups controlling the nondimensional velocity field. Henceforth, we shall refer to the nondimensional group α as the *separation ratio*, since α represents the ratio of the distance between the nozzle exits to the nozzle diameter. Note

that the separation ratio is a parameter describing the most important geometrical feature of a counterflow setup. We hasten to note that this pair of parameters, i.e., (Re_L, α) , is not unique. For example, Re_L may be replaced by $Re_D = 2Re_L/\alpha$ to give (Re_D, α) instead.

A review of analytical,¹⁷ numerical,^{12–14,19} and experimental results^{7,10,15} indicates that the flow field is controlled primarily by the separation ratio and, to a lesser extent, by the Reynolds number. The effect of the nozzle geometry is also apparent, whereby contoured nozzles, rather than straight ones, are adopted in order to inhibit the growth of the boundary layer on the interior walls. Although it is known that the flow field depends on Re and α , issues of similarity, scaling, and dependence from nondimensional groups have not been addressed. Furthermore, quantitative data are lacking and, if available, they are specific to particular setups and of limited scope, so that geometrical effects may not be compared conclusively across facilities and published data.

The objective of our work is to provide a *quantitative* and *comprehensive* characterization of the nondimensional velocity and mixing fields in laminar axisymmetric counterflows in terms of scaling laws. To this end, the parameters Re and α are varied over a wide range of values encompassing most applications in order to explore Reynolds number and geometry effects. Two nozzle types, one straight and one contoured, are considered for completeness. We anticipate that the data originating from our study will aid the design and operation of practical setups, e.g., next-generation counterflow burners for combustion applications at elevated pressures.

The results presented here originate from velocity measurements via Particle Image Velocimetry (PIV), detailed axisymmetric simulations including the nozzle geometry, and exhaustive data reduction based on nondimensional quantities. We report results pertaining to flows with constant density in order to limit the parameter space of our study. The case of variable density flows, which are most pertinent to combustion, will be addressed at a later time.

We describe the experimental and numerical methods in Sec. II. In Sec. III, we present results based on experiments and detailed axisymmetric simulations, which characterize the velocity and mixing fields established between opposed jets conclusively. Attention is paid to those quantities relevant to mixing, such as the scalar dissipation rate. Summary and conclusions are presented in Sec. IV.

II. METHODS

A. Experimental setup and methods

The experimental setup consisted of two opposed contoured nozzles and flow conditioning and metering equipment. The contoured nozzles and flow conditioning stacks are shown in Figure 2(a).

The contoured nozzles were mounted on two identical flow conditioning stacks, consisting of a receiving manifold, a divergent section, and a straight section leading to the inlet of the contoured nozzle (see Figure 2(a)). Two perforated plates with a blockage ratio equal to 0.73 were placed at the inlet and outlet of the divergent section. A honeycomb of height 50 mm and 200 cells per square in. (cpsi) was located in the straight section. The flow conditioning sections (manifold, divergent, and honeycomb) ensure that the velocity at the inlet of the nozzles is uniform, thereby providing well-defined boundary conditions for the simulations (see Section II B). One nozzle assembly was mounted on a flange fastened to the optical table, while the other assembly was mounted on an ensemble of three manual translation stages with micrometer accuracy, allowing for accurate alignment. To improve the alignment further, the stack of translation stages was mounted on a two-axis goniometer.

Three contoured nozzles with exit diameters D equal to 7.5, 15, and 30 mm were used. The flow conditioning units were the same when employing nozzles of different diameters. Nozzles of different sizes were employed in order to support the application of dimensional analysis to the counterflow configuration and rule out additional dependencies from geometrical and flow features other than the nozzle diameter, nozzle separation distance, and bulk velocities (see Sec. III B). Apart from rescaling all physical dimensions by a factor, the geometry of the three nozzles was identical. For the remainder of this paper, the three nozzles will be referred to as S, M, and L, respectively. The shape of the interior surface of the contoured nozzle was inspired by the design in the work

FIG. 2. (a) Overview of the nozzles and flow conditioning stacks: 3D rendering (left) and 2D section (right) from the drawings. (b) The shape of the interior wall of the contoured nozzle (exit diameter 15 mm). All dimensions are in mm. The radii of curvature of the bottom and top portions of the wall are 100.00 mm and 222.66 mm, respectively. The inlet and exit diameters are 57.6 and 15 mm, respectively, resulting in an area contraction ratio of about 15:1. The length of the contoured nozzle is 115.46 mm. The nozzles with exit diameters of 7.5 mm and 30 mm are obtained from the same design rescaled by a factor of 0.5 and 2, respectively. (c) Computational domain and mesh (shown for the case of $H/D = 1$), contoured nozzle geometry, and boundary conditions: axis of symmetry at $r = 0$ (I, blue), no-slip impermeable surface on the nozzle's interior and exterior walls (II, black), uniform velocity inflow and constant mixture fraction (III, red), and mixed inflow/outflow with constant pressure $p_\infty = 1$ atm along a circular boundary located at a radial distance $\approx 12D$ (not shown). Although only one nozzle is shown in the figure for the sake of clarity, the simulations feature two nozzles and the computational mesh is symmetric with respect to the stagnation plane at $z = 0$.

of Rehm and Clemens,²⁰ resulting in an area-based contraction ratio of 15:1. The geometry of the interior wall of the nozzle is shown in Figure 2(b). The contoured nozzles were manufactured using a 3D plastic printer (Dimension BST 1200es, Stratasys). The figure tolerance was found to be equal to ≈ 0.1 mm by direct inspection of the parts.

Nitrogen at 21.6 °C and atmospheric pressure was used as the working fluid ($\rho = 1.16 \text{ kg m}^{-3}$ and $\nu = 1.54 \times 10^{-5} \text{ m}^2 \text{ s}^{-1}$). Two independent feed lines and metering systems were used for the two nozzles. Each line included a mass flow controller (Teledyne–Hastings HFC 303) and PIV seeding unit (LaVision aerosol generator, Item Nr. 1108926). Prior to each use, the mass flow controllers were calibrated with a primary flow calibrator (MesaLabs DryCal Definer 220 primary flow calibrator) resulting in an accuracy equal to $\pm 0.5\%$ of the reading or 1% of the full scale.

PIV was used to measure the axial and radial velocity components on a plane through the axis of the counterflow setup. The PIV system consists of a double-head Nd:YAG laser with a 4 ns, 200 mJ pulse (Litron Laser Nano L200—15 PIV). A camera with 1380×1040 pixels (Imager Intense—LaVision) was used to record the images, which were processed using commercial software (DaVis 8—LaVision).

The time delay Δt between the two PIV images was optimized for each case according to the bulk velocity, the field of view, and the expected displacement of the seeding particles. Depending on the conditions, Δt was between 15 and 450 μs . The two components of the velocity were computed from double frame images using a multi-pass cross correlation algorithm with interrogation windows decreasing from 64 to 16 pixels and an overlap of 50%. The spatial resolution for all cases was between $H/125$ and $H/80$, where H is the separation distance between nozzles. Taking into account the optical properties of the setup and the time separation between pulses, the error for the measured velocity is estimated not to exceed $\pm 6\%$ according to the analysis in the work of Adrian.^{21–23}

B. Numerical methods

The software ANSYS Fluent® (Release 16.0) was used to simulate the velocity, pressure, and mixture fraction fields in the counterflow configuration. The simulations were axisymmetric on a cylindrical coordinate system. Computations are repeated for both contoured and straight nozzles. The computational domain included the internal geometry of the two nozzles and extended up to the nozzles' inlets, corresponding to the exit plane of the honeycomb section in the experimental setup. The computational domain and mesh, nozzle geometry, and boundary conditions are shown in Figure 2(c) for the case of the contoured nozzles. Simulations are repeated with straight nozzles consisting of pipes of length $4D$ also. This arrangement is representative of most counterflow setups with straight nozzles.

At the inlets of the nozzles, uniform axial velocity was prescribed and the mixture fraction was set equal to 1 on one side and 0 on the other. Recall that, in the experimental setup, the contoured nozzles are mounted on a stack of flow conditioning elements (see Figure 2(a)) and the flow out of the terminating honeycomb section is modeled as a uniform inlet velocity profile for the contoured nozzles in the simulation. The interior and exterior nozzle walls were treated as no-slip, impermeable surfaces. On the far field boundary, the pressure p_∞ was set equal to 1 atm.

The boundary layers on the interior walls near the nozzles' exits were discretized with approximately 30 points. Towards the stagnation plane, the grid was refined maintaining an aspect ratio of 1. Starting from the region near the nozzle exit, the grid size was uniform with $\Delta z = \Delta r = H/200$. Moving upstream from the nozzle exit plane towards the nozzle inlet, the grid spacing in the direction normal to the nozzle's inner wall was kept constant, while the spacing in the axial direction was increased, reaching an aspect ratio close to 4. Outside the nozzles, the domain extended radially away from the axis by more than 12 nozzle diameters. The mesh resolution was decreased towards the far field. Overall, the number of cells was approximately 3.5×10^5 .

The grid was found to resolve gradients adequately up to the highest Reynolds number considered. Convergence with respect to the spatial resolution afforded by the mesh was confirmed by repeating the calculations with half the number of grid points without any appreciable modification to the solution.

The momentum term was treated with a second-order upwind scheme. The SIMPLE algorithm was used to couple pressure and velocity. The scalar equation is approximated with a first-order upwind scheme. The equations were solved with the steady solver. Each calculation required approximately 8 h on one core of an engineering workstation (Intel® Xeon CPU E5-2680 v2 2.80 GHz).

III. RESULTS

A. Overview

Figure 3 shows the main features of the laminar and axisymmetric flow field in between two opposing jets issuing from contoured nozzles. The data are reported in a nondimensional form and originate from axisymmetric simulations and PIV-based velocity measurements in a setup equipped with 30 mm contoured nozzles (L). The results of PIV measurements are shown as axial and radial velocity isocontours in Figures 3(a) and 3(b). Due to limited optical access, measurements are provided for the regions $0 \leq z/L \leq 0.9$ and $r/R \leq 1$ only. The pressure field is from the simulations.

The velocity and pressure fields are characterized by the presence of a stagnation plane, which redirects the axial flow issuing from the nozzles outwards in the radial direction. The experimental and numerical data show very good agreement. Flow redirection is associated with a high-pressure region extending axially and radially from the stagnation point. The nondimensional pressure difference $p - p_\infty$ is of magnitude $O(\rho U^2)$ for all cases considered in this study. For atmospheric flows with practical bulk velocities in the range $0.2\text{--}2\text{ m s}^{-1}$, the resulting maximum pressure difference is between 0.05 and 5 Pa with negligible effects on the gas density.

The regions of the flow where viscous forces are important are the thin wall boundary layer of thickness δ_w inside the nozzle and the external shear layer developing between the fast moving

FIG. 3. Overview of the velocity and pressure fields in between two opposing jets issuing from contoured nozzles. The flow parameters are $\alpha = H/D = 1$ and $Re_D = UD/\nu = 1200$ ($Re_L = UL/\nu = 600$). Only the right half of the domain and the top nozzle are shown (fluid moves downward). (a) Nondimensional axial velocity component $-u/U$; (b) nondimensional radial velocity component v/U ; (c) nondimensional pressure difference $(p - p_\infty)/\rho U^2$, where $p_\infty = 1$ atm is the pressure prescribed on the far field boundary. In all figures the color isocontours correspond to data from axisymmetric simulations; in (a) and (b), the solid black lines correspond to PIV measurements carried out in the large contoured nozzle ($D = 30$ mm); in (a), the $u = -U$ isocontour is indicated by a white dashed line; in (b), the $v = 0$ isocontour is indicated by a black dashed line; and in (c), the flow streamlines are shown as black solid lines. (a) $-u/U$. (b) v/U . (c) $(p - p_\infty)/\rho U^2$.

fluid issuing from the nozzle and the quiescent ambient fluid. Vorticity (not shown) is confined to these two regions and is zero everywhere else. With the exception of the wall boundary layer and the external shear layer, viscous forces are negligible everywhere for $Re_D \geq 100$. This behavior is consistent with the flow configuration and the uniform inlet velocity profile, since vorticity is generated at the wall due to shear and is transported towards the centerline, remaining confined inside the wall boundary layer.

Far upstream inside the nozzle, the flow displays the canonical features of an internal flow, which accelerates due to the contraction of the cross-sectional area. The axial velocity is almost uniform across the nozzle and its value increases in the downstream direction (Figure 3(a)). The radial component of the velocity is small and negative (fluid moving towards the centerline) with the exception of the centerline and the wall, where it is zero. The pressure is also nearly uniform radially and decreases towards the nozzle exit to sustain fluid flow.

Moving closer to the nozzle exit, the flow is affected by the presence of the opposing jet and the associated high pressure region centered at the stagnation point (Figure 3(c)). The pressure attains a local minimum and then increases towards the stagnation plane. At the same time, the axial velocity on the centerline decreases, while a region of high axial velocity forms near the nozzle wall, just outside the boundary layer. Corresponding with the local minimum in pressure, the radial velocity becomes positive (fluid moving away from the centerline). In Figure 3(b), the change in the sign of the radial velocity is marked by the $v = 0$ isocontour, which indicates the transition from an internal flow to a stagnation one.

Inside the stagnation flow region, the axial velocity decreases further towards zero, while the pressure and the radial component of the velocity increase. Near the stagnation point, the velocity field conforms closely to that in an *axisymmetric potential stagnation flow* and the strain rate at the stagnation point is the sole governing parameter.

In closing, we remark that the flow is found experimentally to be laminar and steady for all conditions considered in this study, although the Reynolds number based on the nozzle diameter may exceed the critical value of ≈ 2300 that marks the transition from a laminar to a turbulent flow in pipes. This apparent contradiction is explained by noting that Re_D is not representative of the flow regime, since the fluid is accelerated rapidly through the contoured nozzles starting from a uniform velocity profile at the exit of the upstream flow conditioning stacks.

FIG. 4. Radial profile of the nondimensional axial velocity $-u/U$ as a function of r/R at various axial locations. The data are shown for $Re_D = 1200$ and four separation ratios $\alpha = 0.5, 1, 1.5, 2$ (left to right). In all figures, symbols represent PIV data at $z/D = 0.1$ (open circles), 0.2 (solid circles), 0.3 (open squares), and 0.4 (solid diamonds). Solid lines represent simulation results at the same axial location. Dashed lines above and below the solid lines indicate $\pm 6\%$ of the simulated value and provide an indication of experimental errors (see Section II A for the discussion). Colors represent data from the three nozzles: S (green), M (blue), and L (red). In the case of $\alpha = 0.5$, PIV data for $z/D = 0.3$ and 0.4 are not shown, since these axial locations are not optically accessible and the numerical data are reported instead. In the case of $\alpha = 2$, only PIV data from nozzle S are available and shown.

B. Flow similarity

Figure 4 shows the radial profiles of the normalized axial velocity at various axial locations between the stagnation plane and the nozzle exit plane for $Re_D = 1200$. It is apparent that the profiles of the nondimensional components of velocity from the three setups with varying nozzle sizes collapse on each other for given values of the nondimensional groups Re_D and α . Furthermore, the experimental data collapse on the axial velocity profiles from the axisymmetric simulations. Although not shown here, similar observations apply to the radial component of the velocity.

The collapse for the data measured in nozzles L and M is almost perfect, while some differences are apparent for nozzle S, most likely due to challenging conditions for PIV and poor figure tolerances related to 3D plastic printing. Recall that the error on the measured velocity was estimated not to exceed $\pm 6\%$ (see discussion in Section II A). As shown in Fig. 4, both the collapse of the data across nozzles of different sizes and the agreement between experimental and numerical data are within these estimates for the error bounds. Similar trends are found for all other Reynolds numbers considered (not shown).

Additional data on the axial and radial profiles of $-u/U$ and v/U are presented in Figure 5. The axial profile is extracted along the centerline, while the radial profiles are taken at the stagnation plane. Data from PIV measurements and simulations for various Reynolds numbers, separation ratios, and nozzle sizes are reported. Results are shown for $100 \leq Re_D \leq 3000$ and $0.25 \leq \alpha \leq 2$, thereby encompassing all practical counterflow applications in the laminar regime.

It is apparent that the PIV datasets from the three nozzle sizes collapse on each other for a given α and Reynolds number, in agreement with the results shown in Figure 4. Also, the data from the smallest nozzle (S) suffer from the largest measurement errors and inaccuracies, while data from nozzles M and L display a more convincing collapse and agreement with the results of the axisymmetric simulations.

We conclude that the data observed experimentally for the three nozzle sizes confirm the dependence of the nondimensional velocity on α and Reynolds number (either Re_D or Re_L) and validate the application of the Buckingham π theorem to the counterflow configuration as discussed in this work. Recall that the Buckingham theorem does not provide guidance on choosing the relevant dimensional parameters that enter the analysis, so that the selection of such variables amounts to a postulate and is specific to the configuration. Thus, our data, both experimental and numerical, show that the dimensional variables included in the analysis (e.g., U , D , L , and ν) are the *only* variables that govern the flow in between two opposed nozzles and are sufficient to scale the flow entirely.

FIG. 5. (a) Nondimensional axial velocity component $-u/U$ versus z/D along the centerline and (b) nondimensional radial velocity component v/U versus r/R at the stagnation plane. Data are shown for various values of the separation ratio $\alpha = 0.5, 1, 1.5, 2$ (left to right) Re_L : $Re_L = 300$ (open circles), 600 (solid circles), 900 (open squares), and 1200 (solid diamonds). The arrow indicates the direction of increasing Reynolds number. Solid lines represent data from simulations. Colors represent data from the three nozzles: S (green), M (blue), and L (red). In the case of $\alpha = 2$, the data for $Re_L = 300$ are not shown and only PIV data from the nozzle S are available. In each figure, the axial velocity scalings implied by the theories of Chapman and Bauer¹⁶ ($du/dz = -2U/L = -4U/(\alpha D)$) and $dv/dr = U/L = 2U/(\alpha D)$) and Spalding¹ ($du/dz = -2U/D$ and $dv/dr = U/D$) are shown with solid straight lines marked by the labels “C” and “S,” respectively.

C. Separation ratio and Reynolds number effects

Based on the data in Figure 4, it is apparent that the axial velocity displays an important dependence with respect to the separation ratio α . The results shown in Figure 5 support similar conclusions. This dependence is explored further in Figure 6 where detailed data on the axial velocity component for $Re_D = 1200$ and various values of α are shown.

Far upstream in the internal flow region, the flow fields in the three cases are largely identical because Re_D and the wall boundary layer thickness δ_w are the same. A first important difference between the three cases pertains to the spatial distribution of the axial velocity field near the exit plane of the nozzle. As the separation ratio decreases, the $-u/U$ velocity field becomes more inhomogeneous, with a high-velocity region appearing at $r/D \approx 0.5$. Clearly, this region of high axial velocity is related to the slowing down of the fluid near the centerline as is also apparent in Figure 4. Near the stagnation plane, the velocity and pressure fields retain the same qualitative features for all separation ratios.

Along the centerline, the axial velocity $-u/U$ increases moving away from the stagnation point, reaching a maximum at $z/D \leq 1$ for all separation ratios (Figure 5). The radial velocity v/U increases monotonically away from the centerline (Figure 5). The maximum axial velocity along the centerline increases with increasing separation ratio and its location moves further upstream as α increases. For $\alpha = 2$, the maximum is located at the nozzle exit plane. The greatest modifications to the flow occur when the separation ratio increases from 0.5 to 1, while the differences between the velocity fields for $\alpha = 1, 1.5$, and 2 (not shown in Figure 6) are more subtle. This trend is also confirmed by the data shown in Figure 4, where the radial profiles for $\alpha = 1.5$ and 2 are almost identical and very similar to those for $\alpha = 1$. In other words, provided α is large enough, the flow field becomes independent of the separation ratio.

FIG. 6. Contour plots of (a) axial velocity $-u/U$ and (b) pressure $(p - p_\infty)/\rho U^2$ for $Re_D = 1200$ and three separation ratios $\alpha = 0.5, 1, 1.5$ (left to right). Note that the color scale is different in each pressure contour plot for the sake of clarity. In all plots, the thick solid and thick dashed black lines indicate the isocontours $-u/U = 1$ and $v = 0$, respectively. Data from simulations.

The flow field at the exit plane of the contoured nozzle (indicated by a dashed vertical line at $z/L = 1$ in Figure 5) is not characterized by $-u/U = 1$ and $v = 0$: $-u/U \approx 0.6$ for $\alpha = 0.5$, $-u/U \approx 1$ for $\alpha = 1$, and $-u/U > 1$ for $\alpha \geq 1.5$. On the nozzle's exit plane, the axial velocity displays an axial gradient, which is more significant for small separation ratios and decreases for larger separation ratios.

The modifications to the incoming flow field are related to the formation of a high pressure region centered at the stagnation point. It is apparent from Figure 6 that the strength of the high pressure region depends on the separation ratio. The maximum value of the nondimensional pressure difference $(p - p_\infty)/\rho U^2$ is equal to 1.6, 0.7, and 0.5 for $\alpha = 0.5, 1$, and 1.5, respectively. This interaction is governed by the separation ratio α .

Due to the interaction of the two opposing nozzles, the axial velocity at the nozzle exit plane is not uniform, even in the limit of relatively large separation distances and contoured nozzles. These conclusions are in agreement with the trends reported in the work of Sarnacki *et al.*⁷ from PIV-based velocity measurements in counterflow flames for various separation ratios and contoured nozzles.

In Figure 5, axial and radial velocity data are shown for various values of the Reynolds number. For both velocity components, Reynolds number effects are apparent, although not significant. As the Reynolds number increases, $-u/U$ and v/V decrease over the entire axial and radial ranges. This leads to shallower gradients and lower strain rates in the potential flow region and a lower value of the maximum axial velocity away from the stagnation plane. Reynolds number effects are more pronounced for larger separation ratios.

Solutions to the one-dimensional model by Chapman and Bauer¹⁶ and the axisymmetric simulations indicate that the contribution of viscous forces to the velocity balance near the stagnation plane is negligible for $Re_L \geq 100$. We conclude that Reynolds number effects in the counterflow are not due to viscous forces in the stagnation flow region but rather due to the effect of the interior wall boundary layer on the axisymmetric inviscid stagnation flow. As Re_D increases, the wall boundary layer becomes thinner and the axial velocity on the centerline decreases. This mechanism is less important for lower values of α as shown in Figure 5 where the velocity components for $\alpha = 0.5$ do not change appreciably as the Reynolds number of the flow varies.

D. The stagnation point region and the strain rate parameter

The velocity data in Figure 5 indicate that the flow near the stagnation point conforms closely to an axisymmetric potential flow,¹ whereby the axial and radial velocity components vary linearly with the axial and radial coordinates, respectively. In this region, the flow is described uniquely by

FIG. 7. Nondimensional strain rate $k = dv/dr$ at the stagnation point as a function of the Reynolds number for various values of the separation ratio α . Experimental data with the medium (M) and large (L) size contoured nozzles are shown as symbols for $\alpha = 0.5, 1, 1.5$. Thick solid lines represent axisymmetric simulations for contoured nozzles, while thin solid lines represent simulation results for straight nozzles. See Sec. II B for geometrical details. In (a), the strain rate is nondimensionalized with U/D and plotted versus $Re_D = UD/\nu$, and the dashed line (“SPAL”) represents the nondimensional gradient implied by the theory in the work of Spalding.¹ In (b), the strain rate is nondimensionalized with U/L and plotted versus $Re_L = UL/\nu$, and the dashed line (“CB”) represents the nondimensional gradient implied by the model of Chapman and Bauer.¹⁶ The right-hand axis in (b) displays the values of the nondimensional scalar dissipation rate at the stagnation plane $\chi_0 H/U = (4kL/U)/\pi$ according to the one-dimensional mixing solution in the potential flow region (Eqs. (4) and (6) and related commentary for details).

the strain rate parameter k . If we let $k = dv/dr$ indicate the radial gradient of the radial component of the velocity, then $du/dz = -2k$ due to continuity.

Judging by the linearity of the velocity components, the potential flow region extends up to $z/D \approx 0.25$ for all separation ratios (Figure 5). Radially, the region extends up to $r/R \approx 0.3$ for $\alpha = 0.5$ and $r/R \approx 0.5$ for $\alpha \geq 1$ (Figure 5). Our results are in good qualitative agreement with the observations in the work of Liñán,¹⁹ where the potential flow region was reported to extend up to $r/R = 0.5$ in counterflow configurations with $\alpha = 0.5$ and $Re_D > 1000$.

A comprehensive analysis of the strain rate at the stagnation point is presented in Figure 7, where experimental data for contoured nozzles are reported together with results from axisymmetric simulations for both contoured and straight nozzles. For the sake of clarity and completeness, the numerical and experimental data are reported with two different nondimensionalizations, which are consistent with the models of Spalding¹ (Figure 7(a)) and those of Chapman and Bauer¹⁶ (Figure 7(b)). Linear approximations to the axial and radial velocity components according to these two scalings are shown alongside experimental data in Figure 5 also.

The model by Spalding implies that the strain rate k at the stagnation point of a counterflow scales as U/D .^{1,24} This scaling is usually referred to as *potential flow scaling*. The *flow between porous disks model*¹⁶ implies that the strain rate k scales as U/L in the limit of large Re_L when viscous effects at the stagnation plane are negligible. This scaling corresponds to that associated with the *effective strain rate*^{11,15} proposed by Williams⁶ for reactive flows and streams with unequal densities.

1. Dependence on separation ratio

The data in Figs. 5 and 7 document the sensitivity of the strain rate at the stagnation point with respect to departures from ideal inviscid flow fields outside the potential flow region as assumed by either model. In particular, recall that neither the Spalding^{1,24} nor the Chapman and Bauer¹⁶ model includes an explicit dependence of the strain rate at the stagnation plane on the separation ratio α .

The normalized strain rate decreases with increasing Reynolds number for each value of the separation ratio α , regardless of the normalization strategy. The strain rate tends to an asymptotic limit for large values of the Reynolds number and the asymptotic value depends markedly on the

separation ratio α . Another aspect worth noting is that the sensitivity of the normalized strain rate to the Reynolds number is higher at lower Reynolds numbers and larger separation distances. Moreover, the onset of the asymptotic behavior occurs at larger values of the Reynolds number for larger separation ratios α .

Consistently with the data on the velocity components discussed in the commentary to Figure 6, the strain rate displays an important dependence on the separation ratio. Although the dependence of the normalized strain is apparent for either scaling, the behavior differs in the two cases. In Figure 7(a), for a given value of the Reynolds number, the strain rate normalized by U/D decreases monotonically with increasing α . The opposite occurs when the strain rate k is normalized by L/U , so that kL/U increases for increasing values of α (Figure 7(b)). The Spalding theory implies that $kD/U = 1$, while the data in Figure 7(a) indicate that actual strain rates are higher, although the agreement does improve for large values of α and large values of Re_D . For example, for $\alpha = 1.5$ and $Re_D = 1000$, $kD/U \approx 1.3$ and the actual strain rate is about 30% higher than the Spalding limit.

The nondimensional strain rate kL/U implied by the model of Chapman and Bauer¹⁶ is a weak function of Re_L and approximately equal to its asymptotic value $kL/U = 1$ for $Re_L \geq 50$. Unless otherwise noted, the model assumes that uniform axial velocity equal to U and zero radial velocity at a distance L from the stagnation point. This scaling overestimates the actual strain rate at the stagnation point considerably for $\alpha = 0.5$ and $\alpha = 1$, but it happens to be quite accurate for $\alpha = 1.5$ at intermediate values of the Reynolds number. Finally, the strain rate for $\alpha = 2$ is overestimated greatly. The strain rate shown in Figure 7 and the axial and radial velocity profiles in Figure 5 agree with the generally held notion that the strain rate at the stagnation point in an actual counterflow falls between the potential flow¹ and the flow in between porous disks,¹⁶ at least for separation ratios up to ≈ 1.5 .

In the case of the normalization by U/D , shown in Figure 7(a), it is clear that the normalized strain rate kD/U loses its dependence on α for $\alpha \geq 1.5$, while no such limiting behavior is apparent for kL/U in Figure 7(b). Although kD/U loses its dependence on α when the separation ratio is large enough, the data shown for either scaling approach do not support the existence of a limiting behavior with respect to decreasing values of α ($\alpha \rightarrow 0$), at least down to the smallest separation ratio considered ($\alpha = 0.25$).

2. Dependence on Reynolds number

As discussed in the commentary to Figure 5, the sensitivity of the strain rate k to the Reynolds number and separation ratio originates from the effect of those two nondimensional groups on the axisymmetric inviscid flow away from the stagnation point. Recall that viscous effects are localized in the developing wall boundary layer inside the nozzles (see Sec. III A), the thickness of which is controlled by Re_D or $Re_L\alpha/2$. As the wall boundary layer thickness changes in response to changing values of the Reynolds number, so does the centerline velocity, which affects the strain rate k . Higher Reynolds numbers imply thinner boundary layers, resulting in lower centerline velocities and lower strain rates.

It is also apparent that the sensitivity of k to the Reynolds number vanishes with decreasing separation ratio. Thus, as α decreases, the role of the wall boundary layer thickness diminishes and is superseded by that of the high pressure region (see Sec. III B). For small values of α and large pressure differences, the high pressure region controls the incoming velocity field in the transition and stagnation flow regions. Although the pairs (Re_D, α) and (Re_L, α) are equivalent for the purpose of identifying a unique flow configuration ($Re_D = 2Re_L/\alpha$), the fact that normalizing the strain rate by U/D results in an asymptotic behavior of the velocity field for large α suggests that Reynolds number effects in counterflows are parametrized more conveniently by Re_D , rather than Re_L .

3. Dependence on geometry: Contoured vs. straight nozzles

In an effort to quantify further the dependence of our conclusions on the geometry of the contoured nozzles and the role of the interior boundary layer, axisymmetric simulations, albeit not experiments, are repeated for straight nozzles and are shown in Figure 7. Comparisons are

carried out for the same values of Re and α , so that the only difference between the two nozzle geometries is related to contouring, which hinders the growth of the boundary layer on the interior wall.

As shown in Figure 7, for the same values of the nondimensional groups Re and α , our data indicate that the nondimensional strain rates in counterflows with straight nozzles are larger than in those featuring contoured nozzles. This behavior points to higher centerline velocities in straight nozzles compared to contoured ones as a result of thicker wall boundary layers. Moreover, it is clear that differences in the strain rate obtained with the two nozzle designs are small for small separation ratios, becoming more significant as the separation ratio increases.

We conclude that the details of the nozzle geometry are most important for low Reynolds numbers and large separation distances. As the Reynolds number increases, the relative variation of the boundary layer thickness with Reynolds number diminishes ($\delta_w/D \propto Re_D^{-1/2}$), explaining the decreasing sensitivity of the strain rate to Re_D at high Reynolds numbers. Furthermore, the limited sensitivity of the flow field with respect to Re at low values of α is evidence that, for small nozzle distances, the pressure field controls the strain rate at the stagnation point, rather than viscous effects. Our data suggest that comparisons of flow fields across counterflow setups with dissimilar nozzle geometries are best conducted in the limit of small α and large values of the Reynolds number.

E. Mixture fraction and scalar dissipation rate

The steady equation describing the distribution of a conserved scalar with Schmidt number $Sc = \nu/\Gamma = 0.7965$ (Γ is the constant diffusion coefficient) is solved for contoured and straight nozzles. The scalar Z is equal to 0 for one stream and 1 for the other. In the analysis that follows, data from axisymmetric simulations are used since the mixture fraction Z was not measured experimentally.

Figure 8(a) shows $Z(\eta)$ ($\eta = z/L$) for $\alpha = \{0.5, 1\}$ and various Reynolds numbers. The data are extracted from two-dimensional solutions along the centerline. In the stagnation region and outwards for $r/D \leq 1$, the mixture fraction field is one-dimensional and $Z(z, r) = Z(z)$ only, so that the profile along the centerline is sufficient to fully characterize the mixture fraction field. It is apparent that the mixing layer extends across the stagnation plane and is confined to a small region of the domain $-1 \leq \eta \leq 1$. The region of the flow where the two streams mix becomes smaller as the Reynolds number increases. The profiles are such that $Z(\eta) = 1 - Z(-\eta)$ and $Z(0) = 0.5$ due to the boundary conditions $Z(-1) = 0$ and $Z(1) = 1$ and the symmetry of the velocity field.

FIG. 8. (a) Profiles of $Z(\eta)$ ($\eta = z/L$) for $\alpha = \{0.5, 1\}$ and various Reynolds numbers. The data are extracted from two-dimensional solutions along the centerline. (b) Mixing layer thickness $\delta_Z = [\max\{dZ/dz\}]^{-1}$ for $Sc = 0.7965$, normalized by the nozzle separation distance H as a function of the Reynolds number Re_L for various values of the separation ratio α : 0.5 (red squares), 1 (blue circles), 1.5 (green diamonds), and 2 (black pentagons). Data from axisymmetric simulations for contoured and straight nozzles are shown as lines with solid and open symbols, respectively. (c) Compensated nondimensional mixture fraction gradient at the stagnation point, $-Z'(0)/Re_L^{0.5}$, versus the nondimensional (axial) strain rate $-\beta = 2kL/U$. Symbols are as in (b) and the one-dimensional model solution (see Eq. (4)) is shown as a solid line.

Figure 8(b) shows the mixing layer thickness δ_Z , defined as $\delta_Z = 1/\max\{dZ/dz\}$, normalized by the nozzle separation distance H . Since Z is defined between 0 and 1, this corresponds to an equivalent thickness of a linear profile having a slope equal to the maximum slope of the Z profile in the mixing layer.

We hasten to note that, whether the thickness is defined by using scalar levels or as $\delta_Z = 1/\max\{dZ/dz\}$, the scaling of the thickness with relevant parameters remains the same. The maximum gradient of the mixture fraction is preferred here because it provides a compact and convenient analytical expression for the thickness as shown later.

For all conditions considered, $\delta_Z/H \leq 0.1$, indicating that the mixing layer occupies a small fraction of the region between the nozzle exits. If normalized by the nozzle diameter rather than the separation distance, we have $\delta_Z/D \leq 0.1\alpha \leq 0.2$ for $\alpha = 2$.

According to the boundary layer theory² and in the limit of large Reynolds numbers, one has $\delta_Z/H \sim (Re_L Sc)^{-0.5}$. In actual counterflows and for large, albeit finite values of Re_L , the dependence of the strain rate at the stagnation point on α and Re_L causes the mixing layer thickness to show a power law behavior with an exponent that deviates from -0.5 . As shown in Figure 8(b), the difference in the exponent is more pronounced for larger values of α .

The small thickness of the mixing layer across the stagnation plane indicates that mixing occurs inside the potential flow region, where the axial velocity is a function of the axial coordinate z and varies linearly as $u = -2kz$. Upon normalization of the transport equation by the length and velocity scales L and U , one formulates the following ordinary differential equation (ODE):

$$\beta Z' \eta = \frac{1}{Re_L Sc} Z'', \quad (1)$$

where $\eta = z/L$, $\beta = -2kL/U$ is the nondimensional (axial) strain rate, and $(\cdot)'$ indicates differentiation with respect to η . Equation (1) is solved analytically with the boundary conditions $Z(0) = 0.5$ and $Z \rightarrow 0$ for $\eta \rightarrow \infty$ (equivalent to $Z(1) = 0$ for $Re_L Sc \rightarrow \infty$) to obtain¹⁶

$$Z(\eta; Re_L, Sc) = \operatorname{erfc}(\mu)/2, \quad (2)$$

$$\mu = \eta[-\beta Re_L Sc/2]^{1/2}. \quad (3)$$

Due to the form of the similarity variable μ in Eq. (3), an important implication is that, if Re_L is large enough and mixing occurs in a thin region, the nondimensional (axial) strain rate β is the *sole feature* of the velocity field that affects the mixing field. Thus, as far as the mixing field is concerned, the details of the velocity field in regions away from the stagnation plane are relevant only to the extent to which they control the strain rate β .

Based on the solution to the one-dimensional model for the transport of mixture fraction in the potential flow region (Eq. (2)), the nondimensional gradient of Z at the stagnation plane is

$$-Z'_0 = \frac{1}{\sqrt{2\pi}}[-\beta Re_L Sc]^{0.5}. \quad (4)$$

In Figure 8(c), the value of Z'_0 obtained from axisymmetric simulations of the mixing field is compared to that from the model in Eq. (4) and shown as a function of the nondimensional axial strain rate $-\beta$. The reported gradient is compensated by $Re_L^{0.5}$ for clarity. The mixture fraction gradient from the one-dimensional model is identical to that obtained from the axisymmetric model for all separation ratios considered and for both contoured and straight nozzle geometries. This perfect agreement is due to the behavior of the flow field in the limit of high Reynolds number, which confines mixing to the potential flow region.

If we define the scalar dissipation rate as

$$\chi = 2\Gamma(dZ/dz)^2, \quad (5)$$

Eq. (4) leads to the important result that

$$\frac{\chi_0 H}{U} = -\frac{2\beta}{\pi} = \frac{4kL}{\pi U}, \quad (6)$$

where χ_0 is the scalar dissipation rate at the stagnation point, which corresponds to the maximum value of χ in the mixing layer. Equation (6) indicates that the nondimensional scalar dissipation rate

is proportional to β , the nondimensional axial velocity gradient rate at the stagnation plane. The nondimensional scalar dissipation rate $\chi_0 H/U$ is shown in Figure 7(b).

In the limit of large Re_L , the one-dimensional model with boundary conditions $|u(\pm 1)| = U$ implies $\beta \rightarrow -2$ and $\chi_0 = 2U/(\pi L) = 4U/(\pi H)$.¹⁶ Thus, for large enough Reynolds numbers, χ_0 is a function of the *characteristic strain rate parameter* U/L and is independent of the Reynolds and Schmidt numbers, i.e., *independent of the molecular properties of the fluid*.

This limiting behavior makes counterflow configurations well suited for studies on the effect of mixing rates on molecular processes, e.g., chemical reactions in flames. In the limit of high Reynolds number, the characteristic strain rate U/L (or U/H) can be adopted as the *sole governing parameter* that describes the response of the system to mixing, i.e., any Reynolds or Schmidt number dependencies are negligible.

In practice, counterflow setups with nozzles of a given diameter D are operated at constant separation ratios α , and variations in the scalar dissipation rate are obtained by changing the bulk flow velocity U . Thus, as U is adjusted, Re_L also varies and the constant of proportionality linking the scalar dissipation rate to the characteristic strain rate parameter U/L decreases with increasing Reynolds number. As shown in Figure 7, the dependence of $\chi_0 H/U$ on Re_L is important and is far from reaching its asymptotic limit under most practical conditions at low to moderate Reynolds numbers. Consequently, the ratio of the scalar dissipation rate to the parameter U/L is not constant in general, so the response of the system observed experimentally should not be parametrized only by U/L without taking into account the dependence of the velocity and mixing fields on Re_L and α .

IV. SUMMARY AND CONCLUSIONS

We have performed a systematic analysis of the velocity and mixing fields in a counterflow configuration, combining measurements via Particle Image Velocimetry (PIV), detailed axisymmetric simulations including the nozzle geometry. Results have been presented for straight and contoured nozzles.

Our experimental and numerical results indicate that the Reynolds number $Re_D = UD/\nu$ and the separation ratio $\alpha = H/D$ are the sole nondimensional parameters governing the velocity and mixture fraction fields in this configuration. These conclusions are consistent with a straightforward application of dimensional analysis and have been confirmed experimentally and numerically over a broad range of Reynolds numbers ($200 \leq Re_D \leq 4800$), separation ratios ($0.25 \leq \alpha \leq 2$), and for three nozzles of different sizes ($D = 7.5, 15, 30$ mm).

The nondimensional velocity, mixture fraction fields, and the strain and scalar dissipation rate at the stagnation point are found to depend mostly on the separation ratio and, to a lesser extent, on the Reynolds number. At moderate to low Reynolds numbers (e.g., $Re_D < O(100)$), the fields show a dependence on the Reynolds number, which is more significant for larger separation ratios. At high Reynolds numbers, the flow and related quantities attain a limiting behavior which depends solely on the separation ratio. The onset of this limiting behavior occurs at lower Reynolds numbers for smaller separation ratios.

When normalized by the bulk velocity U and the nozzle diameter D , all flow quantities become independent of the separation ratio for $\alpha \geq 1.5$, while there does not appear to be a limiting behavior for the flow fields as the separation ratio α decreases to the smallest value considered, $\alpha = 0.25$.

The strain rate at the stagnation point, defined here as the radial gradient of the radial velocity component ($k = dv/dr$), depends markedly on Re and α . The scaling of k is shown to fall between the Spalding potential flow limit ($k \propto U/D$) and the Chapman and Bauer flow limit ($k \propto U/H$) for the wide range of separation ratios and Reynolds numbers characterized numerically and experimentally.

At moderate to large separation ratios (i.e., $\alpha = 1, 1.5, 2$ in the present study), the thickness of the wall boundary layer on the inner surface of the nozzles affects the inviscid, axisymmetric flow field in between the nozzles by changing the axial velocity away from the stagnation plane. This indirect effect is suitably described by the Reynolds number based on the nozzle diameter,

Re_D . Conversely, at small separation ratios (i.e., $\alpha = 0.25, 0.5$), the flow field is insensitive to the thickness of the wall boundary layer and is instead controlled by the high-pressure region forming at the stagnation point.

By comparing results with contoured and straight nozzles, we observed that the flow field is insensitive to contouring for small separation ratios, but there is a more marked sensitivity of the flow to nozzle geometry at larger separation ratios. The differences in the flow fields obtained with the two types of nozzle become less pronounced as the Reynolds number of the flow increases.

In the range of Reynolds numbers considered ($Re_L \geq 300$), the axisymmetric mixing field exhibits the expected scaling behavior, so that $\chi_0 = 2k/\pi$, where χ_0 is the scalar dissipation rate at the stagnation point. Consistent with the limiting behavior of the strain rate k , $\chi_0 L/U \rightarrow \text{const}$ for large Reynolds numbers, although the asymptotic value of the nondimensional scalar dissipation rate depends on the separation ratio α and differs from the value obtained for the one-dimensional model uniform flow conditions at the nozzle exit planes.

We remark that the normalized scalar dissipation rate reaches an asymptotic limit only for very large Reynolds numbers, e.g., $Re_D \gg 1000$ for $\alpha \approx 1$, so that, in practice, counterflow burners are operated under conditions away from any limiting behavior. Hence the ratio between the scalar dissipation rate and the characteristic strain rate parameter U/L is not constant but is rather a function of the Reynolds number. In simple terms, as the bulk velocity increases for a given geometry, the scalar dissipation rate does not increase at the same rate.

We anticipate that the data originating from our study will aid the design and operation of practical setups, e.g., next-generation counterflow burners for combustion applications at elevated pressures.

ACKNOWLEDGMENTS

The research reported in this publication was supported by the King Abdullah University of Science and Technology (KAUST). Gianfranco Scribano performed part of this work while holding a postdoctoral appointment within the Clean Combustion Research Center at KAUST.

- ¹ D. B. Spalding, "Theory of mixing and chemical reaction in the opposed-jet diffusion flame," *ARS J.* **31**, 763–771 (1961).
- ² H. Schlichting, *Boundary Layer Theory*, 8th ed. (Springer, 2000).
- ³ C. Y. Wang, "Similarity stagnation point solutions of the Navier–Stokes equations—review and extension," *Eur. J. Mech.-B/Fluids* **27**(6), 678–683 (2008).
- ⁴ F. E. Fendell, "Ignition and extinction in combustion of initially unmixed reactants," *J. Fluid Mech.* **21**(2), 281–303 (1965).
- ⁵ F. Carbone and A. Gomez, "Chemical interactions between 1, 2, 4-trimethylbenzene and n-decane in doped counterflow gaseous diffusion flames," *Proc. Combust. Inst.* **35**(1), 761–769 (2015).
- ⁶ F. A. Williams, "A review of flame extinction," *Fire Saf. J.* **3**(3), 163–175 (1981).
- ⁷ B. G. Sarnacki, G. Esposito, R. H. Krauss, and H. K. Chelliah, "Extinction limits and associated uncertainties of nonpremixed counterflow flames of methane, ethylene, propylene and n-butane in air," *Combust. Flame* **159**(3), 1026–1043 (2012).
- ⁸ F. N. Egolfopoulos, N. Hansen, Y. Ju, K. Kohse-Höinghaus, C. K. Law, and F. Qi, "Advances and challenges in laminar flame experiments and implications for combustion chemistry," *Prog. Energy Combust. Sci.* **43**, 36–67 (2014).
- ⁹ A. Alshaarawi and F. Bisetti, "The effect of mixing rates on the formation and growth of condensation aerosols in a model stagnation flow," *J. Aerosol Sci.* **81**, 34–46 (2015).
- ¹⁰ J. C. Rolon, D. Veynante, J. P. Martin, and F. Durst, "Counter jet stagnation flows," *Exp. Fluids* **11**(5), 313–324 (1991).
- ¹¹ H. K. Chelliah, C. K. Law, T. Ueda, M. D. Smooke, and F. A. Williams, "An experimental and theoretical investigation of the dilution, pressure and flow-field effects on the extinction condition of methane-air-nitrogen diffusion flames," *Symp. (Int.) Combust.* **23**, 503–511 (1991).
- ¹² C. E. Frouzakis, J. Lee, A. G. Tomboulides, and K. Boulouchos, "Two-dimensional direct numerical simulation of opposed-jet hydrogen-air diffusion flame," *Symp. (Int.) Combust.* **27**, 571–577 (1998).
- ¹³ V. Mittal, H. Pitsch, and F. Egolfopoulos, "Assessment of counterflow to measure laminar burning velocities using direct numerical simulations," *Combust. Theory Modell.* **16**(3), 419–433 (2012).
- ¹⁴ N. Bouvet, D. Davidenko, C. Chauveau, and L. Pillier, "On the simulation of laminar strained flames in stagnation flows: 1D and 2D approaches versus experiments," *Combust. Flame* **161**, 438–452 (2014).
- ¹⁵ U. Niemann, K. Seshadri, and F. A. Williams, "Accuracies of laminar counterflow flame experiments," *Combust. Flame* **162**, 1540–1549 (2015).
- ¹⁶ T. W. Chapman and G. L. Bauer, "Stagnation-point viscous flow of an incompressible fluid between porous plates with uniform blowing," *Appl. Sci. Res.* **31**(3), 223–239 (1975).
- ¹⁷ K. Seshadri and F. A. Williams, "Laminar flow between parallel plates with injection of a reactant at high reynolds number," *Int. J. Heat Mass Transfer* **21**(2), 251–253 (1978).
- ¹⁸ R. J. Kee, J. A. Miller, G. H. Evans, and G. Dixon-Lewis, "A computational model of the structure and extinction of strained, opposed flow, premixed methane-air flames," *Symp. (Int.) Combust.* **22**, 1479–1494 (1989).

- ¹⁹ A. Liñán, D. Martínez-Ruiz, A. L. Sánchez, and J. Urzay, “Regimes of spray vaporization and combustion in counterflow configurations,” *Combust. Sci. Technol.* **187**(1-2), 103–131 (2015).
- ²⁰ J. E. Rehm and N. T. Clemens, “The large-scale turbulent structure of nonpremixed planar jet flames,” *Combust. Flame* **116**, 615–626 (1999).
- ²¹ R. J. Adrian, “Particle-imaging techniques for experimental fluid mechanics,” *Annu. Rev. Fluid Mech.* **23**(1), 261–304 (1991).
- ²² R. J. Adrian, “Multi-point optical measurements of simultaneous vectors in unsteady flow—A review,” *Int. J. Heat Fluid Flow* **7**, 127–145 (1986).
- ²³ R. J. Adrian, “Dynamic ranges of velocity and spatial resolution of particle image velocimetry,” *Meas. Sci. Technol.* **8**(12), 1393 (1997).
- ²⁴ A. Leclerc, “Déviation d’un jet liquide par une plaque normale á son axe,” *La Houille Blanche* **6**, 816–821 (1950).