PIV measurements of cold flow field in a partially premixed bluff body burner

M. Dutka*, [1], M. Ditaranto[2], T. Løvås[1]

[1]Department of Energy and Process Engineering, Norwegian University of Science and Technology,
Kolbjørn Hejes vei 1B, 7491 Trondheim, Norway
[2]SINTEF Energy Research, 7465 Trondheim, Norway

Abstract

This paper deals with the flow field in a bluff body-based low NO_x laboratory-scale burner. Particle imaging velocimetry (PIV) was used to measure velocity flow field behind the bluff body in the burner. Measurements were conducted for air flow rates ranging from 14 Nl/min to 175 Nl/min. Air speed in the narrowest section of the air duct, i.e. burner throat, varied from 2.7 m/s (Re ≈ 4 900) to 33.8 m/s (Re ≈ 61 500). Non-dimensional length of the recirculation zone is L/D=0,59-0,68. Analysis of the flow field suggests that while the flow behaves quasi-laminar until Re ≈ 4 900, a fully turbulent flow regime for the investigated geometry of the burner starts at Re = 9 800 – 14 700.

Introduction

Bluff body flow aerodynamics has been extensively studied since its significance was recognized by fluid dynamics researchers and engineers in many industries. In the field of combustion bluff body is one of the flame holder strategies which allows for flame stabilization in space and continuous stable combustion process. It plays an important role especially in industrial and practical applications such as boilers and aircraft engines, where gas velocities significantly exceed turbulent flame speeds characteristic for air-fuel mixtures utilized in these applications. Hence, a stabilization mechanism has to be used in order to avoid flame extinction or flame instability. In case of bluff body reactive flow a certain amount of hot combustion products recirculates behind the bluff body and it serves as a heat source for the continuous ignition of the incoming fresh combustible mixture. As a result, the flame is stabilized downstream the bluff body.

Considering that air in the combustion process accounts for approximately 94% of the mass flow, a proper air distribution in the combustion zone is of great importance in order to achieve high performance and low pollutant emissions. As reported in [1], numerous observations have shown that air distribution methods to burners have a significant impact on nitrogen oxides (NO_x) emissions, carbon monoxide (CO) emission and amount of excess air that can be used at stable operation of a combustion system, what in turn translates into achievable energy efficiency.

This paper deals with air flow in a low NO_x laboratory-scale burner. The burner operation is based on rapid fuel and air premixing due to fuel injection into an accelerated air stream and flame stabilization on a bluff body. Motivation for this study is to obtain knowledge of flow regimes in the burner geometry and time-averaged flow pattern for several investigated air flow rates.

The former goal is crucial if one wants to apply dimensional analysis when scaling the burner up to industrial size. It requires obtaining geometric, kinematic and dynamic similarity between the scale model and the prototype, based on similitude theory. However, this is not always possible and the solution to complete dynamic similarity requirement, i.e. to ensure that the ratios of all forces on the fluid flow and geometrical boundaries in the model and the prototype are the same, the principle of relaxation needs to be applied. It means that usually fluid velocity is stressed for the model and the prototype, and the concept of self-similar flow regime is applied [1]. If the flow is self-similar or self-preserving it means that the flow pattern and the pressure drop coefficient are dependent only on the local flow quantities, despite different absolute local flow quantities. This flow regime occurs at sufficiently high Reynolds (Re) numbers, when the pressure drop coefficient becomes independent of the Re number for a given geometry. The studied burner is characterized by a unique design and in order to ensure that the burner operates at a sufficiently high Re number for flow self-similarity assumption, flow regimes characteristic for the burner need to be determined.

The latter goal is important in terms of understanding of the flow pattern in the burner. Flow past bluff body is, in its nature, very chaotic and identification of dominant structures in the flow is challenging. At certain operation settings the burner allows for NO_x emission minimization and it is important to correlate the NO_x emission characteristics with the turbulent field. Time-averaged flow fields allow for quantitative determination of geometrical dimensions of the recirculation zone and comparison of fluid velocities with simplified numerical results of computational fluid dynamics (CFD) analysis, which are performed using Reynolds Averaged Navier-Stokes (RANS) turbulence models in steady state. If acceptable agreement between the experimental and the numerical results are achieved, this approach might be also promising for burner scale-up due to the relatively low computational cost of such simulations. Therefore, knowledge of the flow pattern and the comparison with CFD simulations will be valuable for the burner development.

---

* Corresponding author: marcin.d.dutka@ntnu.no

Bluff body flows

There are differences in characteristics of non-reactive and reactive flows past bluff bodies, but generally characteristic features of the flow remain the same also when there is a flame present downstream the trailing edge of the bluff body.

Even though the flow past bluff body is of inherently unsteady nature, time-averaged flow pattern can be used to analyze characteristic features of the flow. First, boundary layers are formed on the walls of the bluff body upstream its trailing edge. Flow separates from the bluff body at the trailing edge and a recirculation zone is formed behind it. The recirculating wake structures in the recirculation zone are separated from the free stream zone by shear layers, which are characterized by high shear stresses occurring at the interface of these zones. Length of the recirculation zone is determined along the center line and it is the distance between the bluff body surface and the stagnation point. At the stagnation point the streamlines of the free stream converge to the center line closing the recirculation zone [2].

The wakes of bluff bodies typically exhibit an alternating shedding pattern, where the bluff body sheds vortices from the alternating sides [3]. The turbulent shear layers between the recirculation zone and the free stream may experience Kelvin-Helmholtz instability or von Karman instability [4]. These instabilities contribute to vortex shedding phenomenon. These vortices are of unsteady nature and, naturally, they cannot be observed in time-averaged flow characteristics.

Flow pattern behind bluff body depends on many parameters, including geometrical parameters such as bluff body shape and blockage ratio. Researchers studied many different bluff body shapes in various configurations and at various Re numbers. However, most of the studies deal with fundamental processes occurring in the flow field in very simple configurations. Usually a bluff body is placed in a channel flow, where the incoming flow has a uniform velocity profile. The results obtained from these benchmark cases are very helpful in understanding of flow behaviour behind bluff bodies, but if detailed features of the flow need to be analyzed, due to many factors that can affect the flow field, these results cannot be unambiguously translated into other, more complicated geometrical configurations. Therefore, dedicated studies need to be conducted to accurately quantify properties of the double organized and chaotic nature of flows past bluff bodies.

PPBB burner

Bluff bodies can be used to stabilize flames using internal recirculation of hot combustion products. This stabilization mechanism is used in a partially premixed bluff body (PPBB) burner developed and patented at the Norwegian University of Science and Technology (NTNU) and SINTEF Energy Research [5] shown in Fig. 1. It has been shown in [6] that the laboratory-scale PPBB burner exhibits promising low NO_x emission characteristics and it was found that it could be used in refinery fired heaters retrofitted to hydrogen combustion.

The PPBB burner consists of an outer tube terminated by a conically convergent section and an inner tube located concentrically inside the outer tube. The inner tube is terminated by a bluff body, i.e. diverging conical burner head, also called lance, with primary and secondary fuel ports arranged around the burner head. Combustion air flows through the duct between the inner and the outer tubes. High air mass flow rate, compared to stoichiometric fuel mass flow rate, and cross sectional flow area limited by the burner geometry, cause that air stream is the major source of flow momentum in the burner and it has significant impact on the flow pattern behind the lance. Therefore, this is the main reason to study only air flow in the burner geometry.

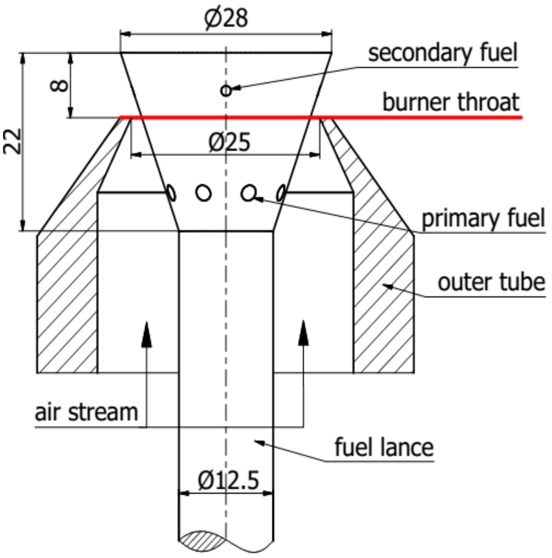


Fig. 1. PPBB burner.

The PPBB burner functions by partially mixing the major part of fuel with air rapidly in the annular space between the outer tube and the lance. This part of fuel is supplied to the suddenly accelerated air stream through primary fuel ports and it is mixed with the air stream in cross-flow. The remaining minor part of fuel is supplied through secondary fuel ports located just before the trailing edge of the lance, wherein these fuel ports are located at a certain distance from the exit of the outer tube. Flame is anchored in front of the burner head due to bluff body flame stabilization mechanism.

Experimental approach

PIV system was used to measure planar velocity flow field behind the burner lance. Only air seeded with olive oil particles was utilized in the experiment and fuel ports located around the lance were not used.

The tested cases are listed in Table 1. Re numbers characteristic for the investigated flows ranged from 4900 to 59 000. Air velocity at the burner throat section and the diameter of the bluff body, i.e. 28 mm, were used for the calculation of the Re number.

High speed Nd: YLF laser and high speed Photron camera synchronized by a PC with Davis 8.2.1 software were used. The setup was arranged as shown in Fig. 2.

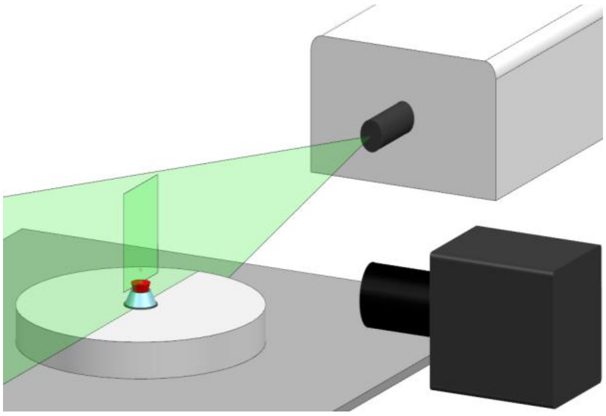


Fig. 2. PIV setup.

Using the PIV system measurements were conducted at a frequency of 1 kHz for 2 seconds, thus 2000 images were acquired per run. A sheet forming lens was used to focus the beam into a thin sheet crossing the axis of the burner lance. Field of view was 107 x 107 mm, giving a pixel resolution of 0.1 mm per pixel.

Davis 8.2.1 software was used to process the PIV vectors. In the PIV algorithm, a multi-pass vector evaluation algorithm was used with the interrogation window size decreasing from 64 x 64 to 32 x 32 pixels with a 50% overlap. This resulted in a spatial resolution of approximately 1.67 mm. Spurious vectors were detected by applying the universal outlier spatial filter. Holes in the processed field left by PIV vector calculation algorithm were interpolated based on velocity vectors in the neighboring interrogation windows. The vector field was smoothed using Gaussian smoothing filter with a 3 x 3 pixel window. Time-averaged flow field was calculated based on the set of instantaneous velocity flow field realizations.

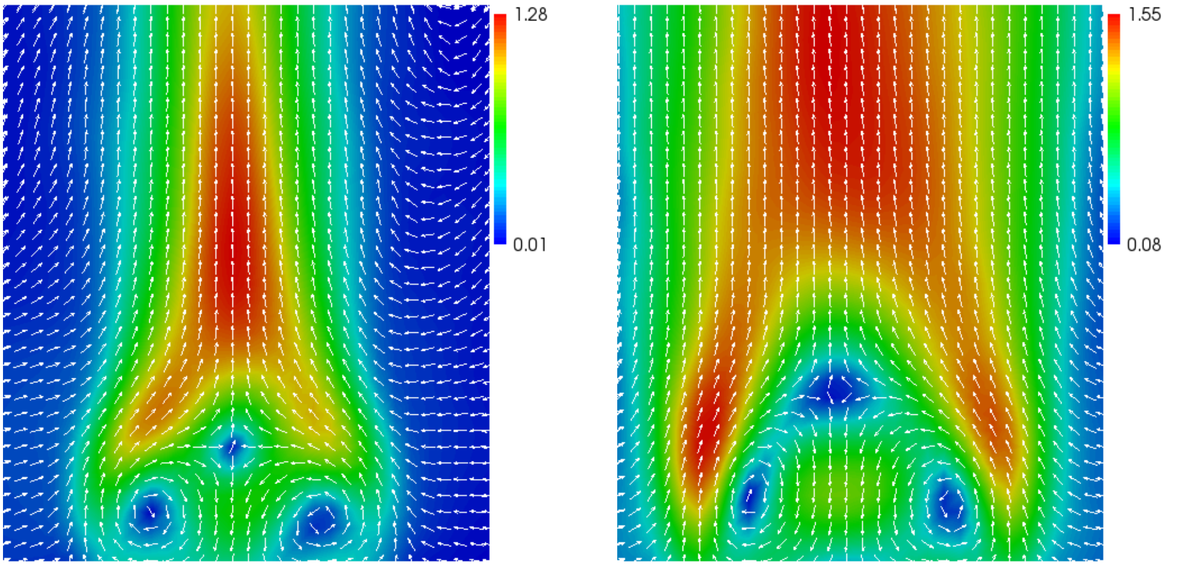
Table 1. Experimental matrix.

Test number	Flow rate [Nl/min]	$v_{\text{burner throat}}$ at normal conditions [m/s]	$v_{\text{burner throat}}$ at laboratory conditions [m/s]	Re [-]
1	14,71	2,5	2,68	4903
2	29,42	5	5,37	9806
3	44,13	7,5	8,05	14709
4	58,84	10	10,74	19612
5	73,55	12,5	13,42	24515
6	88,26	15	16,11	29418
7	102,97	17,5	18,79	34321
8	117,69	20	21,48	39224
9	132,40	22,5	24,16	44127
10	147,11	25	26,85	49030
11	161,82	27,5	29,53	53933
12	176,53	30	32,22	58836

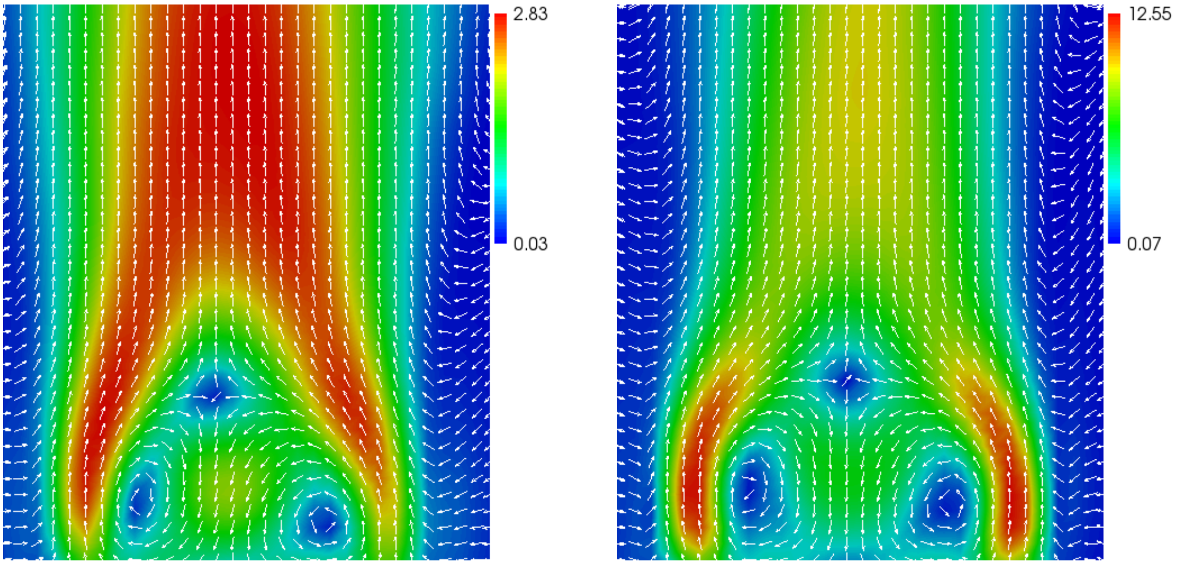
Results

Velocity contours and vectors for all the tested cases are shown in Fig. 2 and Fig. 3. Flow always separates at trailing edge of the bluff body and a recirculation zone is formed downstream, where counter rotating wake structures occur in two-dimensional measurement plane. However, this recirculation zone is toroidal in three dimensions. The recirculation zone is separated from the free stream by the shear layers around the bluff body and the stagnation point, because of high shear stresses caused by opposite directions of the recirculating air and the free stream. At the stagnation point the free stream converges to the centerline of the bluff body. The highest velocities occur at the trailing edge of the bluff body in all the tested cases, except for cases 1 and 2, i.e. Re = 4903 and Re = 9806. A characteristic feature of the flow at Re = 4903 is the fact that the air is accelerated further downstream, what makes this case fundamentally different compared to the other tested air flow rates.

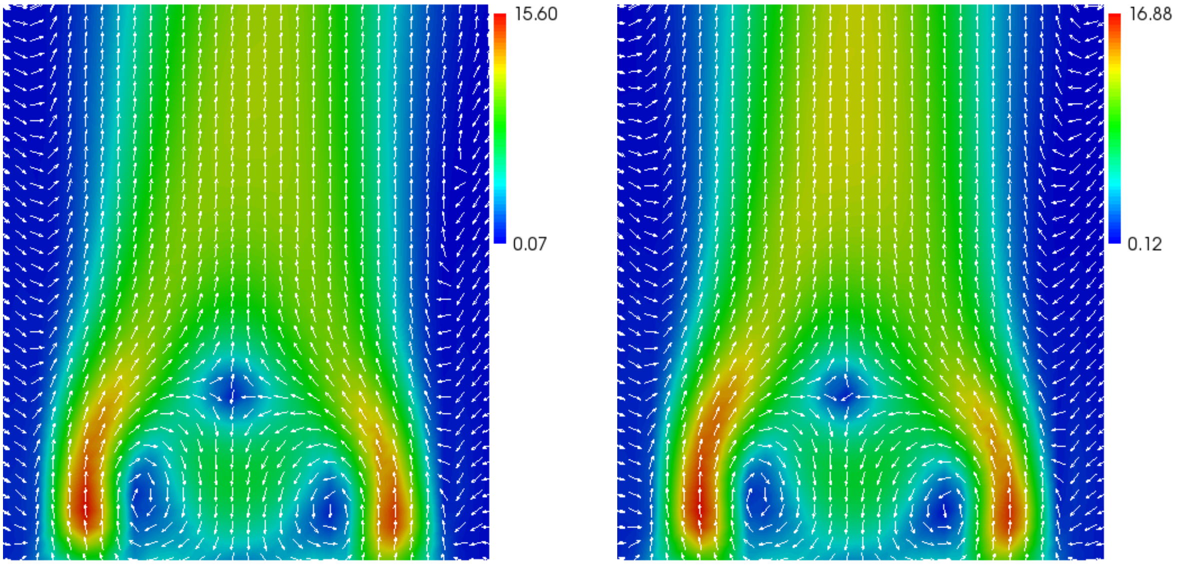
Moreover, observation of the olive oil particles illuminated by the laser sheet, suggests that while the flow is quasi-laminar until Re ≈ 4 900, fully turbulent flow regime for the investigated geometry of the burner starts at Re = 9 800 – 14 700. At Re = 4903 flow is smooth, without any oscillations perpendicular to the flow direction, but when air velocity is increased, i.e. at Re = 9800 and higher, strong chaotic motions and turbulent oscillations were observed.



Case 1 (Re=4903) **Case 2 (Re=9806)**



Case 3 (Re=14709) **Case 10 (Re=49030)**



Case 11 (Re=53933) **Case 12 (Re=58836)**

Fig. 3. Selected time-averaged flow velocity fields at Re numbers ranging from 4903 to 58836. Color intensity represents velocity magnitude in m/s.

In Fig. 4 the lengths of the recirculation zone measured from the bluff body surface to the stagnation point, are presented. The shortest recirculation zone was observed at the lowest tested air flow rate, i.e. Re = 4903 and air velocity at the burner throat equal to 2,68 m/s. In this case the length of the recirculation zone was 12,3 mm. Character of the flow changes significantly with increased Re number and at all the higher flow rates tested in the experiment the lengths of the recirculation zones were in the range 16,5 mm - 18,9 mm. This effect of approach gas speed at high Re numbers on length of the recirculation zone is consistent with findings from other researchers.

However, it is worth noting that in turbulent flow regime measured and non-dimensionalized length of the recirculation zone L/D is of approximately 0,6-0,7, where L is length of the recirculation zone and D is diameter of the bluff body. This is an interesting finding, because it does not agree with the results of other investigations of bluff body flows reported in literature, which usually suggest that length of the recirculation zone is larger than that measured in the present study. In reactive flows recirculation zone closes in the range 2D-4D from a bluff body apex, however, it is known that in reactive flows recirculation zone is larger than in non-reactive flows due to thermal expansion. [2] Nevertheless, even in non-reactive flows lengths of recirculation zone are much larger than that measured in the experiment and achieve 1.24D or more. [7]

In order to explain this result, the effects of various factors on length of the recirculation zone need to be analyzed. It is well-known that length of the recirculation zone is affected by mean flow velocity and bluff body shape [2]. However, not only the burner design is relevant, but wall effects and blockage ratio are important, because, the confinement effect of the side walls has a considerable impact on size and length of the recirculation zone. Furthermore, it was shown in [7] that changing blockage ratio in axisymmetric flows affects the length of the recirculation zone in two ways, depending on what type of flow is investigated. In general, the length of the recirculation zone becomes shorter with increasing blockage ratio, but this applies to unconfined flows, e.g. annular jets, in which the flow is unconfined downstream the bluff body, although the bluff body still presents a blockage. If the flow is confined even further downstream the trailing edge of the bluff body, the size of recirculation zone increases with increasing blockage ratio [8]. For a given blockage and bluff body geometry, the length of the recirculation zone in case of confined flows is larger compared to length of the recirculation zone in unconfined flows.

Even though the concept of blockage ratio cannot be directly applied to the PPBB burner, because of relatively complex design compared to generic cases reported in literature, the above findings can be used to explain the short length of the recirculation zone behind the lance of the PPBB burner.

The PPBB burner consists of an outer tube in which air stream is accelerated due to its convergent shape. This tube can be considered as sort of confinement located upstream the bluff body, which diameter is larger than diameter of the lance, thus the flow area is virtually fully blocked by the lance. Furthermore, the PPBB burner was tested without any chamber or confinement downstream the bluff body. According to the aforementioned discussion on effect of blockage ratio on length of the recirculation zone, these conditions are favorable for shortening the recirculation zone behind the bluff body.

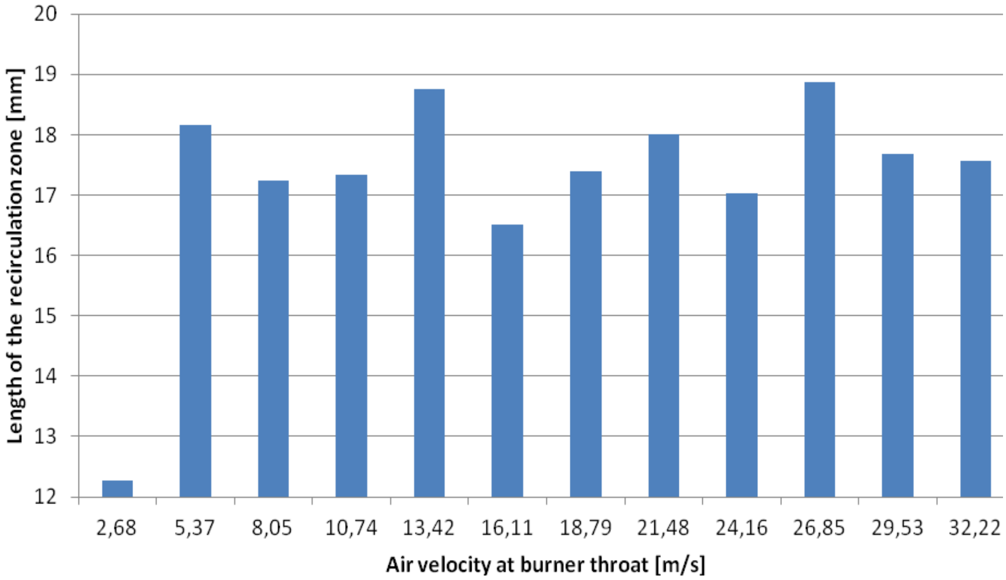


Fig. 4. Length of the recirculation zone versus air velocity at burner throat.

However, the length of the recirculation zone of approximately 1.24D was measured in [9] for flow past conical lance placed in an unconfined channel flow. Also, a similar burner consisting of a conical lance was investigated in [10] and length of the recirculation zone in this paper was reported to be larger than 0,68D.

This suggests that despite the aforementioned conditions resulting in reduction of the recirculation zone size, also the PPBB burner design affects the length of the recirculation zone. Vectors of air velocity at the burner throat are not parallel to the axis of the lance what makes the flow field different compared to most generic cases, where a bluff body is placed in a channel flow with uniform velocity profile of the mean stream. Consequently, the length of the recirculation zone behind the lance is shorter than lengths of the recirculation zones measured elsewhere. This also suggests that the flow field behind a bluff body is very sensitive to the presence of various surrounding walls downstream and upstream the bluff body, to the inlet velocity profile, and more generally to the shape and design of the bluff body.

Conclusions

PIV measurements of the cold flow field in a laboratory-scale partially premixed bluff body burner were conducted. In all Reynolds number cases three characteristic points were observed in the planar PIV measurement plane: two centers of the wakes located behind the bluff body, and a stagnation point. Two flow regimes were identified based on the observation of movement of olive oil particles illuminated by the laser sheet. Analysis of the flow field suggests that while the flow behaves quasi-laminar until $Re \approx 4\ 900$, fully turbulent flow regime for the investigated geometry of the burner starts at $Re = 9\ 800 - 14\ 700$. The length of the recirculation zone, measured from the bluff body surface to the stagnation point, was estimated to be 12,3 mm at $Re = 4903$, while this length changed to between 16,5 mm and 18,9 mm at higher Re numbers, also at the highest tested flow rate at $Re = 58836$. The measured lengths of the recirculation zones were shorter than those reported for generic case study flows past bluff bodies. Important factors affecting the recirculation zone that should be taken into account are: mean flow velocity, blockage ratio, presence of confining walls and inlet velocity profile. The non-dimensional length of the recirculation zone L/D behind the PPBB burner lance is in the range 0,6-0,7 and is significantly shorter than those reported for simple bluff bodies placed in a channel flow. Given the fact that the PPBB burner is characterized by low NO_x emission levels, the information about the size of the recirculation zone is important for the determination of burner operating conditions favorable to NO_x formation minimization and it may be useful for burner scale-up. However, these findings need to be compared with reactive flow field measurements, when a flame is present behind the bluff body. The latter study is part of the on-going research activity.

Acknowledgements

This publication has been produced with support from the BIGCCS Centre, performed under the Norwegian research program Centers for Environment-friendly Energy Research (FME). The authors acknowledge the following partners for their contributions: ConocoPhillips, Gassco, Shell, Statoil, TOTAL, GDF SUEZ and the Research Council of Norway (193816/S60).

References

[1] C.E. Baukal, The John Zink Hamworthy Combustion Handbook, Volume 1, CRC Press, Tulsa, OK, 2013

[2] J. A. Lovett, C. Cross, E. Lubarsky, B.T. Zinn, A review of mechanisms controlling bluff-body stabilized flames with closely-coupled fuel injection, Proceedings of the ASME Turbo Expo, 2011

[3] A. Roshko, On the Development of Turbulent Wakes From Vortex Streets, PhD thesis, California Institute of Technology, Pasadena, CA, 1952

[4] A. Prasad, C.H.K Williamson, J. Fluid Mech., 333 (1997) 375-402.

[5] Ø. Spangelo, O.K. Sønju, Y. Slungaard, M. Ditaranto, Method for burning of gaseous fuel and burner, Patent numbers: NO324171B1, US8292615B2

[6] M. Ditaranto, R. Anantharaman, T. Weydahl, Energy Procedia 37 (2013) 7214-7220.

[7] A.M.K. P. Tylor, J.H. Whitelaw, J. Fluid Mech., 139 (1984) 391-416

[8] G. Winterfeld, Proc. Combust. Inst., (1965) 1265–1275

[9] T.W. Davies, J.M. Beer, Proc. Combust. Inst., (1971) 631-638

[10] D. Dovizio, M.M. Salehi, C.B. Devaud, Combust. Theor. and Modelling, 17 (2013) 935-959