

Ortolani, A., Yeadon, J., Ruane, B., [Paul, M.](#) and [Campobasso, M. S.](#) (2023) Numerical and experimental analysis of the cold flow physics of a non pre-mixed industrial gas burner. *Journal of Fluids Engineering*, 145(8), 081202. (doi: [10.1115/1.4062165](https://doi.org/10.1115/1.4062165))

Reproduced under a Creative Commons License.
<https://creativecommons.org/licenses/by/4.0/>

Copyright © 2023 by ASME

This is the author version of the work. There may be differences between this version and the published version. You are advised to consult the published version if you want to cite from it:
<https://doi.org/10.1115/1.4062165>

<https://eprints.gla.ac.uk/295162/>

Deposited on 27 March 2023

1
2
3

Numerical and experimental analysis of the cold flow physics of a non pre-mixed industrial gas burner

4 **A. Ortolani***

Ph.D. Student

Department of Engineering

University of Lancaster

Lancaster, United Kingdom LA1 4YW

Email: a.ortolani@lancaster.ac.uk

J. Yeadon

Proctor Process Plant Ltd.

Burnley, United Kingdom BB11 5UB

5 **B. Ruane**

Proctor Process Plant Ltd.

Burnley, United Kingdom BB11 5UB

M. Paul

Professor

School of Engineering

University of Glasgow

Glasgow, United Kingdom G12 8QQ

Email: manosh.Paul@glasgow.ac.uk

6 **M.S. Campobasso**

Senior Lecturer

Department of Engineering

University of Lancaster

Lancaster, United Kingdom LA1 4YW

Email: m.s.campobasso@lancaster.ac.uk

*Corresponding author

ABSTRACT

The flow field of a non-premixed industrial gas burner is analyzed with Reynolds-averaged Navier Stokes computational fluid dynamics validated against velocity and pressure measurements. Combustion is not modeled because the aim is optimizing the predictive capabilities of the cold flow before including chemistry. The system's complex flow physics, affected by a 90° turn, backward and forward facing steps, and transversal jets in the main stream is investigated at full and partial load. The sensitivity of the computed flow field to inflow boundary condition set-up, approach for resolving/modeling wall bounded flows, and turbulence closure is assessed. In the first sensitivity analysis, the inflow boundary condition is prescribed using measured total pressure or measured velocity field. In the second, boundary layers are resolved down to the wall or modeled with wall functions. In the third sensitivity analysis, the turbulence closure uses the $k - \omega$ shear stress transport eddy viscosity model or two variants of the Reynolds stress model. The agreement between the predictions of most simulation set-ups among themselves and with the measurements is good. For given type of inflow condition and wall flow treatment, the ω -based Reynolds stress model gives the best agreement with measurements among the considered turbulence models at full load. At partial load, the comparison with measured data highlights some scatter in the predictions of different patterns of the flow measurements. Overall, the findings of this study provide insight into the fluid dynamics of industrial gas burners, and guidelines for their simulation-based analysis.

Keywords: Industrial gas burner fluid dynamics, Navier-Stokes Computational Fluid Dynamics, Reynolds-stress and $k - \omega$ SST turbulence models, Pressure and velocity measurements

25 1 Introduction

26 Turbulent combustion sits at the interface of the chemistry and turbulence disciplines, both characterized by a key
27 role of nonlinearity and multi-scale phenomena. This makes the simulations of turbulent combustion processes based on
28 Computational Fluid Dynamics (CFD) a challenging task, even when the chemical mechanisms at work are well known. For
29 example, this is the case of methane-air diffusion flames, which have been the focus of intense research for several years. In
30 this field, many simulation studies have focused on relatively simple laboratory experiments in well controlled conditions,
31 such as the Sandia flames [1–3]. Despite the apparent simplicity of this reactive flow, the prediction of some of its features,
32 including the formation of pollutants such as nitrogen oxides, remains a difficult task. These modeling challenges may be
33 higher when using the Reynolds-Averaged Navier Stokes (RANS) approach [4,5], rather than the higher-fidelity Large Eddies
34 Simulation (LES) approach [6, 7]. This is because a necessary prerequisite for reliable predictions of turbulent combustion
35 is a sufficient resolution of flow turbulence, an aim achievable more easily by using LES, which resolves the larger scales
36 of turbulence. Unfortunately, the computational burden of LES is higher than that of RANS simulations, and therefore LES
37 is often not amenable to industrial applications. On the other hand, a wide fidelity spectrum also exists in the domain of
38 RANS methods, and the RANS simulation outcome is also affected by several modeling choices. Thus, it is of interest to
39 keep improving the RANS methodology for turbulent combustion analysis and burner design.

40 Most industrial burners are characterized by geometries and flow fields which are far more complex than those of
41 reference laboratory experiments, such as the Sandia Flames. In industrial applications, it is important to optimize and
42 validate the predictive capabilities of the CFD analysis of the cold flow of the system, ie that without chemistry, before
43 including also reactive flow modeling in the simulations. This is because the levels and patterns of flow turbulence have a
44 strong impact on the predictions of the combustion process [8]. Recently, this approach was adopted by Wronski et al. [9],
45 who analyzed the cold flow field of a magnesium burner performing RANS analyses with the CFD code ANSYS FLUENT.
46 To avoid handling simultaneously the uncertainty affecting the analysis of swirling flows in geometrically complex ducts, and
47 that associated with modeling the combustion of the two-phase flow of magnesium and air, they started their investigations by
48 modeling the cold one-phase swirling air flow. The authors compared the results of their simulations with experimental data
49 that they obtained for two operating conditions, characterized by different levels of flow swirl. They tested several variants
50 of the $k - \epsilon$ turbulence model [10] and of the Reynolds Stress Model (RSM) [11]. It was found that, for the low-swirl case,
51 the Renormalization Group $k - \epsilon$ turbulence model [10] performed slightly better than the other variants. In the high swirl
52 case, the RSM variant based on the ω -equation of the standard $k - \omega$ [12] gave the best predictions. In both cases, the shape
53 and position of the zone with negative values of axial velocity in the main duct could be predicted with reasonable accuracy.

54 Meraner et al. [13] conducted a numerical study of the cold flow in a partially premixed bluff body burner, and compared
55 their computed velocity fields with the Particle Image Velocimetry measurements of Dutka et al. [14]. The authors tested
56 three RANS eddy viscosity turbulence models, namely the standard $k - \epsilon$ model, the realizable $k - \epsilon$ model [15] and the $k - \omega$
57 Shear Stress Transport (SST) [16]. An overall good agreement with experiments was found, particularly with regard to the
58 size of the flow recirculation region behind the bluff body. However, the predicted magnitude of the axial velocity deviated
59 from the experimental data. The $k - \omega$ SST model performed better than the other two models in capturing the velocity

60 decay in the jet region downstream of the recirculation zone, but worst in terms of predicting the overall velocity level. The
61 agreement between SST-based predictions and experiments improved when the analysis was carried out in unsteady mode.
62 Even larger improvements were observed using LES and a stress-blended eddy simulation (SBES) [17, 18].

63 The general aim of this study, which is part of a wider ongoing research, is to develop an experimentally validated
64 computationally affordable RANS CFD technology for the analysis and design of industrial gas burners. The objective of
65 the analyses herein is two-fold: on one hand, it is to investigate and shed light on the complex fluid dynamics of an industrial
66 gas burner, supporting the findings on its flow physics with measurements of its flow field; on the other hand, the objective
67 is to present parametric analyses of the simulation set-up, including inflow boundary condition (BC) choice, approach to the
68 solution of wall-bounded flows, and turbulence closure, and provide guidelines on the best choices in RANS CFD simulations
69 of industrial gas burners. The investigation focuses on the nonreactive flow of the burner because of the importance of an
70 adequate prediction of turbulent flow patterns to reliably predict turbulent combustion problems, as discussed above. The
71 considered test case is a non-premixed industrial burner for natural gas and methane combustion. The burner is designed to
72 operate in continuous industrial processes with a firing range from 12 to 120 KW. The flow simulations and measurements
73 of this study refer to two load conditions using only air as working fluid. The main novelty of this study is the investigation
74 of the cold flow physics of a non-premixed industrial gas burner, and its dependence on the operating condition. Since the
75 control of the turbulent flow pattern is one of the means available to improve the efficiency and reduce the emissions of this
76 system, predicting and explaining the key flow features is paramount to its design optimization. The experimental part of this
77 investigation is carried out by using a full-scale test rig that reproduces the conditions in which the gas burner is operated in
78 production. This makes the presented analyses relevant to both the scientific and industrial communities of this sector.

79 The outline of the paper is as follows. Section 2 presents the test rig and a general description of the fluid flow paths
80 in this case study. In Section 3 the experimental set-up and the procedure followed for the measurements are described.
81 Section 4 describes the CFD code and methodology, whereas Section 5 defines the physical domain, grids, and BCs. Section
82 6 assesses the grid independence of the CFD solutions. Section 7 presents the results of this study: first, the main findings of
83 the parametric analyses varying inlet BC, calculation of the wall-bounded flows, and turbulence models are presented; then
84 the main features of the flow field of the system are presented and discussed. Finally, Section 8 provides a summary of the
85 study with comments on future work.

86 **2 Test Rig**

87 The outer geometry of the considered burner, consisting of a case containing part of the nozzle, is reported in Fig. 1a.
88 A fan, connected to the burner with an inlet duct, provides the air flow supply (Fig. 1b). The fan works at constant angular
89 speed, with the air flow rate being regulated by a throttling valve located just before the case. The inlet duct has a rectangular
90 cross section equal to that of the throttling valve case. Figure 1b also shows the combustion chamber bolted to the burner
91 case. The chamber has a cylindrical shape and it is open on the outlet section, communicating directly with the external
92 ambient. Figure 2a shows that the nozzle consists of a conical part surrounded by a coaxial cylinder. The cone has several
93 holes arranged in a periodic pattern. Figure 2a also highlights a flame detector, and a spark plug. Together with some bolts,

94 these two components are the only elements breaking the axial symmetry of the nozzle geometry.

95 [FIGURE 1]

96 The schematic of Fig. 2b shows the flow path in the burner. The stream of fresh air enters the case, and after a 90° turn,
97 it splits into multiple co-axial streams. The innermost stream flows in a tube ("nozzle inlet duct") whose axis lays on the
98 centerline of the chamber. This is the stream indicated by the central blue arrow. This stream then passes through the nozzle
99 and reaches the combustion chamber. Most of the remaining air enters the chamber through the holes on the conical part of
100 the nozzle. Secondary air streams enter in the combustion chamber following two different paths: the small gap between the
101 external face of the cylinder and the burner case, and the gap between the internal face of the cylinder and the cone. When
102 the burner is firing, the fuel stream follows the path of the red lines in Fig. 2b, guided by the fuel system ducts shown in
103 Fig. 2a. The fuel mixes with the air directly in the nozzle, and in this way, it feeds the flame.

104 [FIGURE 2]

105 Since this study focuses on the cold flow field of the considered system. Therefore, the fuel inlet is disconnected from
106 the fuel supply. It is also sealed, so that no air enters from there. In this way, the fuel inlet box communicates only with the
107 nozzle via the fuel duct shown in Fig. 2a.

108 Figure 3 provides the symbols used in this study to denote the characteristic lengths of the system. The diameter and the
109 length of the cylindrical combustion chamber, not reported in Fig. 3 for brevity, are denoted by d_c and l_c , respectively. Table
110 1 reports all characteristic lengths normalized with the nozzle external diameter d_n , which equals 125mm.

111 [FIGURE 3]

112 [TABLE 1]

113 3 Experimental set-up

114 The measured quantities in this study are static pressure, total pressure, and velocity. Flush mounted pressure taps are
115 used to measure the static pressure. The probe used to measure the total pressure consists of a tube with a hole whose axis
116 is oriented as the incoming flow. Both static and total pressures are measured with the SDP816-500PA analog transducer
117 by Sensirion [19]. This sensor measures the difference between probed and ambient pressures, and it outputs an analog
118 ratiometric voltage. The sensor covers a range of $\pm 500Pa$ with an accuracy of $\pm 3\%$ of the measured value $\pm 0.1Pa$. Five
119 of these transducers are connected to an Arduino board and collect 500 samples of static or total pressures with a sampling
120 frequency of $5.4Hz$. In this study, the static and total pressures measured with these transducers are labeled "SDP816".

121 Some static and total pressure measurements are repeated with a second differential manometer to verify the calibration
122 of the SDP816 transducer. For this purpose, the Kimo MP 200 P manometer [20], characterized by a range of $\pm 500Pa$
123 with an accuracy equal to $\pm 0.2\%$ of the measured value $\pm 1.5Pa$, is used. This manometer measures the difference between
124 probed and ambient pressures, and outputs its value on a screen. As this sensor does not have data-logging capability, an
125 average pressure value is computed with 10 screen readings. In this study, the static and total pressures measured with

126 this manometer are labeled "Kimo". Unless otherwise stated, the measured pressure data reported below are the SDP816
127 transducer.

128 A constant-temperature thermal anemometer, namely the TA440 model by TSI Incorporated [21], measures the velocity
129 along the desired direction. The measured velocity component depends on the orientation of the probe. The measuring
130 range of this sensor goes from 0 to $30\frac{m}{s}$ with an accuracy of $\pm 3\%$ of the measured value and a resolution of $0.01\frac{m}{s}$. The
131 anemometer has a sampling frequency of $1Hz$. This sensor measures time-averaged values of the velocity over $1s$. Each
132 measurement is carried out for more than $5min$, providing velocity time-histories consisting of over 300 samples.

133 All measured time-series are elaborated to compute the mean value and the Root-Mean-Square (RMS) of the deviations
134 from such mean value at each measurement point. The measured quantities presented in the following sections, unless
135 otherwise stated, are in form of time-averaged values.

136 The measurement locations are divided in three subsets or *stations* to help the discussion. Each station consists of all the
137 measurements carried out in one region of the test rig. Station 1 is located at the mid length of the inlet duct, and is depicted
138 in Fig. 4. Here the sensors are used to measure velocity and total pressure with the aim of providing data for the inlet BCs of
139 the CFD simulations. Both quantities are measured at five positions, indicated by red dots in Fig. 4. The orientation of the
140 probes is such that they capture the component of the flow velocity in the Y direction. The X and Z velocity components are
141 neglected. The static pressure is measured on the mid point of each side of the measuring section (blue dots in Figure 4).

142 [FIGURE 4]

143 Figure 5 provides a schematic of the station 2, showing the locations at which the static pressure is measured on the
144 burner case. A pressure tap is positioned on the fuel box (FB), as its pressure provides an indirect measure of the velocity in
145 the nozzle itself. This method allows collecting information on the flow in the nozzle in a non-intrusive way. The figure also
146 shows lines F1 and F4 along which the static pressure on the burner case is measured.

147 [FIGURE 5]

148 A schematic of station 3 is given in Fig. 6a. The axial component of the velocity (V_x) is measured in the combustion
149 chamber, namely along the three transverse lines E1, E2, and E3. Figure 6a also shows the origin of the selected reference
150 frame. The origin is on the nozzle and chamber centerline, and its X position is at the bottom of the combustion chamber.

151 All the measurements are repeated for two load conditions. One is that of full load, when the throttling valve is fully
152 open (condition V90). The other is the partial load regime, corresponding to a partial valve opening of 50° (condition V50).
153 A schematic of the valve orientation for the two configurations is shown in Fig. 6b.

154 [FIGURE 6]

155 4 Computational fluid dynamics method

156 All numerical analyses are carried out with the commercial CFD code ANSYS FLUENT [22], using version v19.3,
157 unless otherwise stated. The code is used to solve the pressure-based incompressible RANS equations. FLUENT uses the

158 finite volume method for the space discretization of all conservation laws. In all analyses, a second order upwind scheme for
159 the convective fluxes is chosen. Diffusion terms are discretized with second-order finite-differencing. All simulations herein
160 are time-dependent, and, all reported numerical results are time-averages of unsteady CFD simulations using a first order
161 discretization scheme in time with a dual time-stepping approach. None of the considered simulations could be performed
162 using a steady state solver, because, even if not large, the physical level of unsteadiness prevented convergence to a mean
163 state to be achieved with a steady solver. All simulations use the COUPLED solver, that solves the continuity and momentum
164 equations in a strongly coupled fashion, whereas all other transport equations are solved in a loosely coupled fashion.

165 Part of the presented analyses uses the $k - \omega$ SST turbulence model [16]. The model is based on Boussinesq hypothesis,
166 and computes the turbulent viscosity μ_t from the turbulent kinetic energy k and the specific turbulence dissipation rate (ω),
167 which are transported variables. The remainder of the analyses uses the higher-fidelity RSM approach [11].

168 RSM is a RANS approach that is not based on the Boussinesq hypothesis, and better accounts for the anisotropy of
169 turbulence. Thus, it is often better suited for cases where this character is more pronounced, such as highly swirling flows
170 [9, 23]. The independence of the Reynolds stress tensor τ_{ij} on the laminar stress tensor in the RSM approach, requires
171 solving a transport equation for each of the six distinct components of τ_{ij} . Moreover, an additional transport equation for the
172 dissipation of τ_{ij} needs to be solved. Therefore, RSM uses 7 transport equations to model turbulence. This increases notably
173 computational costs with respect to two-equation turbulence models.

174 Several options are available for the equation of the τ_{ij} dissipation. These can be subdivided in ϵ -based methods, where
175 ϵ is the turbulent dissipation rate, and ω -based methods. Two variants of each approach are implemented in FLUENT. The
176 ϵ -based variants differ for how they model the pressure strain term in the τ_{ij} equations. The default option is that proposed
177 by Gibson and Launder [24], Fu et al. [25], and Launder [26, 27]. This solution, named "linear pressure strain term", is less
178 accurate than the "quadratic pressure strain term" by Speziale, Sarkar, and Gatski [28], but is found to be more stable. The
179 ϵ -based RSM variant tested in this study is the latter one; it has been found that, in order to prevent these simulations from
180 becoming numerically unstable, the convection terms of the transport equations of τ_{ij} and ϵ have to be discretized with a first
181 order upwind scheme.

182 The available ω -based RSM variants are the RSM- ω and the RSM-BSL variants. The RSM- ω model is based on the
183 ω -equation of the standard $k - \omega$ model of Wilcox [12], which was shown to give free-stream sensitive results [29]. The
184 RSM-BSL model uses instead the ω -equation of the baseline $k - \omega$ model of Menter [30], which removes the free-stream
185 sensitivity. The ω -based RSM variant tested in this study is the latter one, which unlike the tested ϵ -based variant has been
186 found to be sufficiently stable also with a second order upwind discretization of the convective terms of turbulence transport
187 equations.

188 The RSM simulations are carried out with FLUENT version v21.2, as the use of v19.3 led to numerical instabilities
189 causing residuals to rapidly grow, and the simulation to crash after just a few time steps.

190 5 Numerical set-up

191 5.1 Physical domain and grids.

192 The physical domain considered herein is shown in Fig. 7. It starts at the measurement station 1, it includes throttling
193 valve, burner case and nozzle, and it contains the cylindrical combustion chamber. Preliminary analyses highlighted the
194 necessity of resolving also the flow field around the combustion chamber to avoid strong numerical instabilities caused by
195 recirculation regions reaching the outlet of the chamber. Thus, the physical domain extends $85d_c$ downstream of the chamber
196 outlet, and $37.5d_c$ radially, as indicated in Fig. 7. Figure 7 shows the main outer boundaries of the physical domain, which
197 include the primary inlet through which the fan feeds the burner, the secondary inlet through which air flowing past the
198 combustion chamber enters the domain, the outlet boundary through which the primary and secondary air leave the domain,
199 and the outer boundary. The BCs applied on these boundaries are stated in Section 5.2. The physical domain does not include
200 the flame sensor, the spark plug and the bolts.

201 [FIGURE 7]

202 All meshes are generated with ANSYS FLUENT Meshing, and are high-quality hybrid unstructured grids with regular
203 hexahedral cells in most of the domain. Polyhedral cells are used close to the boundaries, in regions connecting grid portions
204 with different refinement, and where the complexity of the geometry requires their use. Figure 8 shows the longitudinal
205 section of a grid with medium refinement for the V90 operating condition for simulations which resolve wall boundary layers
206 (BLs) down to the wall without wall functions. As the number of cells increases significantly when solving BLw down to the
207 wall, two different approaches have been assessed in this work. One reduces the computational cost by generating an inflation
208 layer with only two cells in the direction normal to the walls. This method used wall functions (WFs). The other approach
209 resolves BLs down to the wall. This is enabled by generating an inflation layer that guarantees a nondimensionalized wall
210 distance y^+ of the cell centers closest to the walls smaller than 1 almost everywhere. For the fluid problem studied herein,
211 the overall number of grid cells for the same level of grid refinement on the interior domain doubles when BLs are resolved.
212 This method is labeled "WR" in the remainder of this article. Section 7.1 will present a comparative analysis of the results
213 obtained with the WR and WF.

214 [FIGURE 8]

215 5.2 Boundary Conditions

216 The BCs on the primary inlet boundary are prescribed as 2D maps of either total pressure or velocity. The two cases are
217 labeled "P" and "V", respectively. These maps are obtained elaborating the data measured at station 1.

218 The spatial variations of the measured total pressure at station 1 are relatively low. The maximum deviation from the
219 mean of the 5 values measured at the 5 positions indicated by red dots in Fig. 4 is less than 2.5%. Including also the measured
220 wall static pressure (blue dots in Fig. 4) this difference increases to 16%. Since these variations are relatively small, and
221 the highest differences arise in a narrow region close to the wall, the prescribed total pressure is based only on the measured
222 points indicated in red. A constant interpolation method is used, whereby the total pressure on a point of the boundary, is the

223 closest measured value. The result of this operation is shown in Fig. 9a, in the form of total pressure profiles extracted from
224 the generated maps.

225 The spatial variations of the measured velocity in the primary inlet duct are more significant than those of the total
226 pressure, with a maximum deviation from the mean above 15%. Moreover, the velocity decreases sharply near the walls,
227 becoming zero there. Figure 9b shows the velocity profiles for the CFD simulations generated from the measured profiles.
228 The CFD input values are interpolated along the lines $x_b = 0$ and $x_h = 0$ (Fig. 4 using a shape-preserving cubic Hermite
229 interpolation. They are then extrapolated linearly to a distance from the wall of 20% of l_{x_b} and l_{x_h} , respectively. The profiles
230 are then extended to the wall with the logarithmic law of the wall [31]. The 2D velocity map on the primary inlet boundary
231 is obtained by interpolations based on the two profiles described above.

232 [FIGURE 9]

233 All wall boundaries, except the domain outer boundary, are treated as viscous walls. An inviscid wall condition is instead
234 applied on the domain outer boundary (Fig. 7. A pressure outlet condition, which enforces zero gauge pressure, is enforced
235 on the outlet boundary of the domain. To avoid the occurrence of back-flow on this boundary, a non-zero V_x of 0.002 m/s is
236 on the secondary inlet boundary imposed secondary inlet boundary.

237 Choosing either of the aforementioned methods for prescribing the BC at the primary inlet, and either of the approaches
238 described in Section 5.1 for handling wall BLs, four possible set-ups are defined, labeled "P-WF", "P-WR", "V-WF", and
239 "V-WR" in the parametric analyses of Section 7.1.

240 6 Mesh sensitivity analysis

241 Selected results of a three-mesh sensitivity study are presented in this section. Due to the high computational cost of
242 these time-dependent simulations, the analysis is carried out only for the P-WF with $k - \omega$ SST model for the V90 operating
243 condition. The limiting factor for the set-up choice is the computational cost of the fine grid simulation. The WF set-up was
244 selected because the WR fine grid has a larger cell count of 17.43 million, and, more importantly, the convergence rate of the
245 time-averaged flow field to a steady state was found to decrease when reducing the cell size in the near-wall regions. Both
246 factors made the use of a fine grid without wall functions not affordable with the available computational resources. The
247 reason for selecting the $k - \omega$ SST rather than an RSM set up, was also to make the computational burden of the fine grid
248 simulation affordable.

249 The coarse, medium, and fine grids have, respectively, 3.8, 5.9, and 8.7 million cells. For reference, the WR grid with the
250 same refinement of the medium WF grid away from walls has 12.0 million cells. Since the simulations are time-dependent,
251 only the time-averaged flow fields computed with the three grids are compared. A constant time-step of $dt = 5 \cdot 10^{-3} s$ with
252 25 iterations at each time step are used in all simulations. The flow-through time, an estimate of the time required for a
253 fluid particle to travel from the primary inlet boundary to the end of the combustion chamber, is defined as the ratio of a
254 characteristic length and velocity. The characteristic length is defined as the sum of three lengths: the combustion chamber,
255 the burner case and portion of the inlet duct included in the physical domain. The considered characteristic velocity is

256 evaluated as the average measured velocity at station 1. All simulations are initialized with a hybrid initialization, and have
 257 been run for about 10 flow-through times in order to achieve a statistically stationary condition. From this time, simulations
 258 are run for another 50 flow-through times, and a time-average solution over this time interval is obtained at the end of the
 259 simulation. The solution sensitivity to mesh refinement is assessed by comparing local and global values of the mean flow
 260 field computed on the three grids. One considered parameter is the computed mass flow rate \dot{m}_{air} . Another parameter used
 261 for the analysis is the RMS of the differences between the local velocity V_x in a section A normal to the X axis and the mean
 262 velocity V_{bc} in the same section. The definition of this global metric is:

$$RMS^A = \sqrt{\frac{\int_A (V_x - V_{bc})^2 dA}{A}} \quad (1)$$

263 All velocities in Eq. (1) are final time-averaged values of the simulation, and, therefore, RMS^A provides only a measure of
 264 the spatial variability of the velocity, and not a measure of possible unsteady fluctuations.

265 The first, second and third rows of Tab. 2 report integral quantities computed on the coarse, medium and fine refinement
 266 grids, respectively. The second column provides the mass flow; the third, fourth and fifth columns provide the value of
 267 RMS^A , respectively, on the cross sections at the longitudinal positions where lines L1, L2, and E3 are positioned in Fig. 6a.
 268 One sees that the values of the mass flow rates are very similar in all three grids, and the three values are within about 0.4% of
 269 each other. The value sets of RMS^A at the three axial positions highlight that the differences between medium and fine grids
 270 are notably lower than those between the medium and the coarse grids in all cases. In fact, the RMS^A percentage differences
 271 between medium and fine grids lay between 0.8% and 1.9%, whereas those between coarse and medium grids range from
 272 2.0% to 7.5%. These data provide a first indication of sufficient grid independence of the medium grid.

273 [TABLE 2]

274 The profiles of V_x computed on the lines L1, L2 and E3 are reported in Figures 10a, 10b and 10c, respectively. Figure
 275 10b reports also the profiles of the radial velocity component V_r , which is significant at this position, due to the transverse
 276 jets in the conical nozzle. At all three positions, the profiles obtained with the medium and fine grids differ notably less than
 277 those obtained with the coarse and medium grids. On the line L2, the differences between coarse and medium profiles of
 278 V_r are particularly significant at $\frac{Y}{d_n} \approx \pm 0.3$, where the peaks of the radial velocity component due to the jets are observed.
 279 On line E3 in the combustion chamber, large differences between the coarse and medium profiles of V_x are also observed,
 280 despite the smaller flow gradients in this region.

281 [FIGURE 10]

282 Figure 11 compares the coarse, medium and fine grid wall pressure profiles on the line F1 indicated in Fig. 5. Consis-
 283 tently with the trends highlighted above, the fine and medium grid profiles are superimposed, whereas the coarse grid profile
 284 differs slightly from the other two for $\frac{X}{d_n} < -1.5$. That is the region where the flow from the primary inlet duct hits the facing
 285 wall of the burner case, causing static pressure to increase at this location.

[FIGURE 11]

The results above indicate the suitability of the medium refinement grid to properly resolve the flow physics of the considered industrial gas burner. As stated above, three wall BL resolving grids could not be used for the complete mesh sensitivity analysis, due to the computational burden of the fine grid analysis. However, the comparison of the results obtained with a BL-resolving 8.44 million-cell coarse grid and a 12.00-cell medium grid, not reported for brevity, showed relatively small differences of the two analyses, also in the near-wall regions. This occurrence gives confidence that also the medium level of refinement of WR set-up is adequate for the scopes of the reported analyses.

7 Results

The first part of this section presents a parametric analysis comparing a selection of results obtained with the numerical set-ups P-WF, P-WR, V-WF, and V-WR defined at the end of Section 5. In the second part, the solutions obtained with three turbulence models using the same wall BL approach and inlet BC type are compared. Finally, the detailed analysis of the flow field of the analyzed system is presented in the third subsection. The section also presents comparisons of the simulations with all available measured data.

7.1 CFD solution sensitivity to inlet BC and wall BLs resolution method

The $k - \omega$ SST turbulence model is used in all four numerical set-ups obtained by using either primary inlet BC and either wall BLs solution approach discussed in Section 5.

Figure 12a shows three sets of V_x profiles in the combustion chamber for the operating condition V90. The axial positions E1, E2, and E3 to which the profile sets refer are those indicated in Fig. 6b, and each set reports the CFD profiles computed with the four set-ups and the measured profile. The error bars on the experimental profiles are the RMSs of the deviations from the mean values of the measured time-series at each measurement point. The measurements at some locations were repeated, and the results are reported on the same plots. The left plot of Fig. 12a shows that, although the magnitude and position of the peak velocity are well predicted by P-WF, the shape of the profile differs from those predicted by the other three set-ups, which are closer to the measured data. This is a first indication of a poorer predictive performance of the P-WF set-up. At the axial positions E2 and E3, the Y position of the P-WF V_x peak is different from that of all other profiles (middle and right plots). Further investigations not reported for brevity show that a likely cause is that using wall functions to model part of the BL around the throttling valve results in a flow reversal where the valve has maximum thickness. This separation, not present when the BL is resolved down to the wall, affects the vortical patterns in the burner case and, subsequently, the V_x profiles in the combustion chamber. The flow reversal in the valve region is present also with the set-up V-WF. However, in the V-WF solution, its impact on the downstream flow is reduced due to the convective forces which, in the valve region, are higher than those of the P-WF flow field. This is because the velocities prescribed in set-up V-WF are higher than those computed with P-WF.

[FIGURE 12]

318 The P-WR, V-WR, and V-WF velocity profiles are close to each other. The only significant difference is seen in the left
 319 plot of Fig. 12a where the peak velocity is underpredicted by P-WR. The profiles predicted by the set-ups with prescribed
 320 velocity at the primary inlet differ very little, suggesting that the prediction improvements associated with resolving BLs for
 321 the V90 regime are small when the primary inlet velocity is prescribed.

322 Figure 12b refers to operating condition V50, and has the same structure as Fig. 12a. The P-WF set-up appears to per-
 323 form better than in the V90 case, since its predictions are now closer to those of the other three set-ups and the experimental
 324 data. The left plot of Fig. 12b highlights that the profiles computed prescribing the inlet velocity differ significantly only
 325 in the central part of the profile, for $-0.4 < \frac{Y}{d_n} < 0.2$. Significant differences occur instead over most part of the profiles
 326 at stations E2 and E3 (middle and right plots). This suggests that these differences may be due primarily to differences in
 327 the flow field of the nozzle inlet duct. Thorough flow field investigations reveal that these differences occur in two regions:
 328 where the flow accelerates past the valve, and where it enters the nozzle inlet duct generating a recirculation region in the
 329 duct itself. These phenomena will be explained in detail in Section 7.3.

330 Table 3 reports the mass flow rate \dot{m}_{air} estimated using the experimental data and the results of the P-WF and P-WR
 331 simulations for the two operating conditions. The \dot{m}_{air} values computed by integrating the 2D velocity maps based on the
 332 measured velocities are reported in the Exp_{est} column. Those computed by the two CFD simulations are reported in the
 333 columns labeled P-WR and P-WF. The percentage differences of the \dot{m}_{air} values of two simulations are reported in the Δ_{CFD}
 334 column. The P-WF estimates are between 5.4% and 6% larger than P-WR estimates. These notable differences underline
 335 the importance of using the more reliable wall BL resolving approach rather than wall functions to properly correlate mass
 336 flow rate and pressure jump. The values of \dot{m}_{air} estimated from the measured velocities are consistently higher than those
 337 computed by CFD. This may be due to the uncertainty affecting the generation of the 2D velocity maps at the primary inlet.

338 [TABLE3]

339 The static pressure analyses below use the nondimensionalized pressure p^* , defined by Eq. (2).

$$p^* = \frac{p}{p_{t0}} \quad (2)$$

340 where p is the local static pressure and p_{t0} is a reference pressure corresponding to the total pressure at the center point of
 341 the primary inlet boundary. Both p and p_{t0} are gauge pressures. The measured values of p_{t0} , and those computed by the
 342 V-WR and V-WF set-ups are provided in Tab. 4 (the P-WR and P-WF set-ups enforce the measured p_{t0} value. Since in the
 343 V-WR and V-WF set-ups, p_{t0} is an output of the simulation, it provides an indication of the computed pressure jump when
 344 the velocities are prescribed at the primary inlet. The measured p_{t0} values are fairly reliable, as indicated by the fact that the
 345 Kimo manometer p_{t0} value regime V50 is 421.9Pa, very close to the value measured by the SDP816 transducer in Tab. 4.

346 [TABLE 4]

347 Figures 13a and 13b report the results obtained for the V90 regime on the lines F1 and F4, respectively, highlighted in

348 Fig. 5. Figure 13a shows that the trend of the P-WF profile differs slightly from that of the other three set-ups. The cause
349 of this deviation is likely to be the same yielding the P-WF velocity patterns observed in Fig. 12a and discussed above. The
350 profiles of p^* predicted by V-WF are lower than those predicted by the other three set-ups, but the slope of this profile is
351 similar to that of the P-WR and V-WR profiles. Figure 13b underlines that the V-WF profile on line F4 is lower than the
352 other three numerical profiles. The results obtained for the V50 regime, not reported for brevity, show that all CFD profiles
353 are very close to each other.

354 [FIGURE 13]

355 Finally, Tab. 5 reports the values of p^* evaluated in the fuel box (FB in Fig. 5) for all considered cases. The value of
356 p_{FB}^* measured with the Kimo transducer for the V50 condition is -0.069 . V-WR and P-WR seem to perform slightly better
357 than V-WF and P-WF, even though the differences are minimal. This may suggest that resolving wall BLs has a stronger
358 impact on the nondimensionalized static pressure predicted in the fuel box than the choice of the quantity prescribed at the
359 primary inlet.

360 [TABLE 5]

361 The results shown in this section indicate that P-WF gives poorer predictions in full load conditions, while V-WR over-
362 estimates velocity magnitudes in partial load conditions. Thus, P-WR has been chosen as baseline set-up for the remainder
363 of the analyses, as it gives good predictions in both operating conditions and is more trustworthy than V-WF which does not
364 resolve BLs.

365 7.2 CFD solution sensitivity to turbulence model.

366 This section presents a parametric study on the impact of using either the SST or the RSM turbulence model in the
367 P-WR analysis of the considered problem.

368 Figures 14a and 14b, featuring the same structure of Figures 12a and 12b, report the V_x profiles on in the combustion
369 chamber for operating conditions V90 and V50, respectively, using the considered turbulence models. At design conditions
370 (Fig. 14a), the two RSM set-ups predict very similar velocity profiles. The RSM set-ups yield slightly better prediction
371 than the $k - \omega$ SST, especially in the region close to the nozzle exit (line E1). Some larger differences between the two RSM
372 set-ups occur for the V50 condition Fig. 14b). The RSM-BSL model predicts the position of the peak velocity and the overall
373 shape of the distribution better than both the SST and RSM- ϵ . The latter two models, however, yield a better prediction of
374 the peak velocity magnitude on the E1 line. The $k - \omega$ SST model performs slightly worst than the RSM variants, especially
375 in the V90 condition, where the values of V_x that it predicts close to the nozzle exit are too low with respect to measurement
376 (left plot of Fig. 14a).

377 [FIGURE 14]

378 The comparison of the p^* profiles on the F1 and F4 lines, not reported for brevity, does not show significant differences
379 among the predictions using the three turbulence models.

380 Table 6 provides the values of p^* in the fuel box. One sees that all models succeed in predicting a small negative gauge
381 pressure, differing from the experimental value by less than 1% of p_{t0} for the $k - \omega$ SST and RSM-BSL set-ups. The RSM- ϵ
382 model underestimates the magnitude of the FB gauge pressure, particularly at design conditions. A possible cause may be
383 the use of a first order discretization for the convective terms in the τ_{ij} and ϵ equations, as discussed in Section 4. Overall, all
384 turbulence models perform fairly well for the V50 condition, although the RSM predictions are closer to the experimental
385 measurement than the $k - \omega$ SST model. The p^* value recorded by the Kimo manometer for the V50 regime is -0.069 ,
386 slightly different from the reading of that SDP816 transducer, and very close to the RSM-BSL prediction.

387 [TABLE 6]

388 The comparative analysis above highlights that, overall, the RSM-BSL set-up yields prediction improvements over the
389 $k - \omega$ SST and the RSM- ϵ set-ups. Therefore, the RSM-BSL method is used in the detailed flow analyses presented below.

390 7.3 Flow field analyses

391 In the analyses below, the P-WR RSM-BSL set-up is used to investigate the key flow features of the gas burner for both
392 V90 and V50 operating conditions.

393 7.3.1 Flow field at design conditions

394 Figure 15 shows the flow field in the burner case at the design conditions V90. As the valve is parallel to the direction
395 of the oncoming flow, the air stream flows around the valve without flow reversals. However, separation occurs downstream,
396 due to the backward facing steps at the junction of the inlet duct and the burner case. The resulting vortical structures are
397 visible in all three subplots of Fig. 15b. The recirculation zone labeled "A₁" covers the whole width of the burner case. The
398 fuel duct and plate, highlighted in purple and green, respectively, in Fig. 15b, and visible more clearly in Fig. 2a, interact
399 with the counter-rotating vortex in the upper region of the case, breaking it into two weaker vortices labeled "B₁" and "B₂".
400 The three plots of Fig. 15b are similar to each other, and show that the velocity gradients in the X direction are small in
401 this region. In the region highlighted by the orange rectangle in Fig. 15a, one sees that the fluid accelerates entering the
402 nozzle inlet duct. A recirculation region is formed on the inner wall of the nozzle inlet duct opposite to the direction of the
403 oncoming flow.

404 [FIGURE 15]

405 Figure 16 reports the velocity field in the YZ planes whose X positions are indicated in Fig. 15a. In all three planes, the
406 asymmetry of the V_x contour maps of the flow approaching the nozzle from outside the fuel duct is visible. Flow asymmetry
407 exists also in the nozzle inlet duct, as clearly visible in Fig. 16a. The reduction of V_x in the Y direction indicates that most
408 of the flow rate goes through the half of the burner case at $Y < 0$. This effect is particularly significant at the section closer
409 to the inlet duct (left plot). Figure 16 also illustrates the downstream development of the vortices identified in Fig. 15b. The
410 strongest vortex "A₁" dominates the flow in the burner case. The vortex "B₂" is completely dissipated, as it is no longer

411 visible in Fig. 16b. The vortex "B₁" does not disappear completely, but its intensity decreases significantly as the stream
412 advances in the burner case.

413 [FIGURE 16]

414 The velocity field of the nozzle is analyzed in Fig. 17. A fast stream on the centerline, originating from the nozzle inlet
415 duct, is visible in Fig. 17a. The velocity contour map also highlights the asymmetry of this primary stream. High-speed
416 secondary streams or jets emanate from the holes on the nozzle cone, and merge with the primary stream. Figures 17b and
417 17c show the flow patterns before and after the nozzle. One can see that the vortex "A₁" generated upstream is still significant
418 at the inlet of the combustion chamber.

419 [FIGURE 17]

420 Figure 18 shows the contour plot of p^* in the plane $Z = 0$ of the burner case. The figure shows a stagnation region
421 (highlighted by an orange rectangle) resulting from the impingement of the oncoming flow entering the case from the inlet
422 duct.

423 [FIGURE 18]

424 Computed and measured p^* profiles on lines F1 and F4 are compared in Figures 19a and 19b, respectively. On line
425 F1, both experiments and simulations predict higher static pressure in front of the inlet duct ($\frac{X}{d_n} < -1.5$) than immediately
426 downstream ($\frac{X}{d_n} > -1.5$). This higher pressure is caused by the impingement of the oncoming primary flow on the wall of
427 the burner case. On line F4 (Fig. 19b), both simulations and experiments show that p^* is almost constant, and a small adverse
428 pressure gradient exists. Both plots of Fig. 19 show that the level of static pressure in the burner case is overpredicted by
429 CFD. This implies that the pressure drop in the inlet duct and past the valve is underestimated by CFD. This could be due to
430 the fact that the computed velocity field on the primary inlet boundary does not feature transverse velocity components, and,
431 therefore, does not take into account all the expected three-dimensionality of the flow coming from the fan.

432 [FIGURE 19]

433 Measured and computed profiles of the V_x velocity component on lines E1, E2, and E3 are compared in Fig. 20. In-
434 spection of these profiles indicates that the lack of axial symmetry in the burner case and around the nozzle extends to the
435 combustion chamber. The measured profiles of all plots of Fig. 20 show that the peak velocity is shifted towards positive Y
436 values, and this pattern is correctly predicted by the CFD simulation. The asymmetry is due to the highly 3D flow field in the
437 burner case, caused primarily by the 90° turn when the primary stream enters the burner case. Computed velocity profiles
438 close to the nozzle (left and middle plots) are in good agreement with experiments both in terms of peak velocity and shape
439 of the profile. Further downstream (right plot) the computed V_x profile is slightly higher than the measured profile, although
440 the overall agreement remains fairly good.

441 [FIGURE 20]

7.3.2 Flow field at off-design conditions

Figure 21 presents the velocity field in the burner case for the off-design regime V50. As highlighted in Fig. 21a, the direction of the longer axis of the valve cross section forms an angle of 40° with the oncoming flow direction. Due to this high flow incidence to the valve, a low-speed recirculating flow region forms behind the valve. A second flow feature caused by this orientation of the valve is an acceleration of the two streams flowing on the two sides of the valve. These high velocity streams are highlighted with yellow ellipses. The stream highlighted by the lower ellipse reaches directly the nozzle inlet duct in the region indicated by an orange rectangle. When this stream hits the inlet area of the nozzle duct, a separation bubble forms on the inner wall of the duct facing the oncoming stream, similarly to the V90 operating condition. Figure 21b shows the presence of two secondary flows in the Z direction at the end of the inlet duct, highlighted by purple arrows in the central plot. These secondary flows are due primarily to the two forward facing steps located at the top and bottom of the inlet duct, indicated by green circles in all three subplots. To a minor extent, vortices A_1 and B_1 also contribute to the formation of these secondary flows, by pushing the flow towards the center of the inlet duct. The point where the two secondary flows meet is marked by a red dot in Fig. 21a. This flow feature also exists at the design condition V90, but, due to the greater momentum of the flow in the Y direction following the valve (i.e. larger V_y level), it is less pronounced. This can be seen by comparing Figures 15b and 21b. The comparison also highlights that the velocity variations in the X direction, in this region of the burner case, are stronger in the V50 condition. Figure 21b also shows the presence of two secondary vortices, " A_0 " and " B_0 ", which are not observed at design conditions. These vortices are not convected downstream: they are visible in the top subplot of Fig. 15b, and by the time the flow reaches the section at $X = X_3$, they are no longer visible.

[FIGURE 21]

The three subplots of Fig. 22 examine the velocity field at the axial stations X_4 , X_5 and X_6 indicated in Fig. 21a. The behavior of the flow in this region is similar to that of the V90 regime considered in Fig. 16, with some differences. For example, the interaction of vortices A_1 and B_1 , which have different relative strengths with respect to the design condition, lead to the formation of a small secondary vortex B_3 at station X_5 , not observed in the V90 operating condition.

[FIGURE 22]

The flow field in the nozzle is visualized in Fig. 23. The overall velocity level is in the V90 condition. Figure 23c also shows that the vortex A_1 persists in the combustion chamber, similarly to the design condition.

[FIGURE 23]

Figure 24 compares CFD and measured p^* profiles along lines F1 and F4, and its inspection leads to similar considerations to those reported in Section 7.3.1 for the V90 condition. For the V50 operating condition, two sets of measured static pressure are available, one measured with the SDP816-500PA analog transducer (Exp-SDP816 in the legend) and one measured with the Kimo MP 200 P manometer (Exp-Kimo in the legend). In this operating condition $p_{t0} = 424.7\text{Pa}$ for SDP816 and CFD, while $p_{t0} = 421.9\text{Pa}$ for Kimo. Figure 24 shows that the two transducers give very similar readings. The simulation predicts a peak static pressure at $-2 < \frac{X}{d_n} < -1.5$ not seen in the experimental data. The CFD peak occurs at

475 the position where the stream highlighted by the yellow ellipse below in Fig. 21a hits the wall of the burner case. Possible
476 reasons for this mismatch could be that the position and size of the high pressure region may depend on seemingly minor
477 geometric features not included in the physical domain. Moreover, the distribution of the pressure taps may be too coarse to
478 resolve this pressure variation. Both the computed and measured pressure profiles on line F4 (Fig. 24b shows that the adverse
479 pressure gradient in the streamwise direction is stronger than that observed in Fig. 19b for the V90 operating condition. The
480 magnitude of the predicted and computed pressure gradient is in good agreement.

481 [FIGURE 24]

482 Measured and computed V_x profiles in the combustion chamber are compared in Fig. 25. An overall good agreement of
483 experimental measurements and numerical results is observed. At the axial position E1 (left plot) close to the nozzle, a fairly
484 good agreement is observed, with some discrepancies arising only in the region around the centerline ($Y/d_n = 0$), where
485 the numerical model overpredicts the measured V_x profile. The agreement improves further moving downstream, as visible
486 in the middle and right plots, comparing measurements and simulations at positions E2 and E3, respectively. Comparing
487 the three subplots of Fig. 25 to those of the V90 regime reported in Fig. 20, highlights that also in the V50 condition all V_x
488 profiles have a positive Y offset, indicating a lack of axial symmetry of the flow in the combustion chamber.

489 [FIGURE 25]

490 8 Conclusions

491 The main features of the cold flow physics of a non-premixed industrial gas burner at full and partial load have been
492 investigated by means of RANS CFD, and flow measurements taken in a full scale test rig have been used for CFD validation.
493 Parametric CFD analyses aiming at assessing the impact on the computed solution of *a*) inflow BC type (imposed velocity
494 map or total pressure), *b*) resolution of wall-bounded flows (wall functions or integration down to walls), and *c*) turbulence
495 closure ($k - \omega$ SST, RSM-BSL or RSM- ϵ models) have been carried out.

496 At both operating conditions, the flow field is dominated by highly 3D flow phenomena, including: *a*) a strong deviation
497 of the flow field in the burner case until downstream of the nozzle exit from the axisymmetric pattern, due to a 90° turn of the
498 flow between the air admission duct and the burner, *b*) a system of large secondary vortices caused by the abrupt change in
499 cross sectional area at the end of the air inlet duct, *c*) a separation bubble at the beginning of the nozzle inlet duct, due to the
500 flow arriving from the air admission duct being orthogonal to the nozzle inlet duct, and *d*) flow recirculation pockets caused
501 by forward and backward facing steps on the inner walls of the entire system. In the partial load condition, the overall flow
502 field pattern is made more complex also by the stalled flow pocket around the admission valve, and the two air jets between
503 the valve ends and its bounding walls.

504 Overall good agreement of CFD results and experimental data has been observed at both operating conditions using
505 the RSM-BSL turbulence model with imposed total pressure at the inlet of the air admission duct, and integration of the
506 governing equations down to the wall to resolve the near-wall flows.

507 The analysis of the solution sensitivity to the resolution of the wall bounded flows and the inflow BC type, carried out

508 using the $k - \omega$ SST model, show that, at the full load operating condition, the largest deviations from the measured data are
509 observed when using wall functions and imposing the inlet total pressure. At the partial load condition, the four solutions
510 are relatively close to each other and in fairly good agreement with measured data.

511 Comparative analyses of the four solutions of the full load condition, indicate that the deviation of the solution with
512 imposed inlet pressure and using wall functions from the other three solutions may be due to both the inadequacy of the wall
513 function approach to handle flow separation, and the sensitivity of the velocity field to local flow separations being larger
514 when imposing the total pressure rather than the velocity at the inlet of the air admission duct. As the observed flow separations
515 occur only in localized regions, one could refine the near-wall grid and resolve BLs only in separated flow regions, and use
516 wall functions elsewhere. This hybrid set-up would enable achieving reduced computational costs and overall adequate
517 solution fidelity.

518 The improvement of the agreement of simulations and measurements obtained by using RSM may not seem sufficiently
519 large to justify its increased computational cost over that of two-equation eddy viscosity models. However, available literature
520 shows that RSM is better suited than eddy viscosity models to predicting turbulent diffusion flames. This holds also for gas
521 burners notably simpler than the industrial gas burner considered herein, for example purely cylindrical combustors with
522 fuel and oxidizer forming two coaxial non-swirling flows [8, 32]. Therefore, the RSM-BSL closure is deemed to be a well
523 suited method for the follow-on analysis of the reactive flow of the considered burner. Nevertheless, burner design studies
524 have also highlighted the potential benefits that optimizing certain burner flow patterns may have on improving combustion
525 efficiency and reducing pollutants [33, 34]. In design optimization, which typically requires analyzing a large number of
526 system variants, computationally more affordable eddy viscosity RANS set-ups also play an important role in the initial
527 phase of burner design optimization.

528 Acknowledgements

529 This project is supported by the Center for Global Eco-Innovation, the European Regional Development Fund, and
530 Proctor Process Plant, which designed and manufactured the industrial burner. All the simulations in this work were run on
531 the Lancaster University High End Computing (HEC) cluster. The authors are grateful to the Reviewers for their helpful
532 comments, and the Editors for handling the article submission.

533 References

- 534 [1] Barlow, R., and Frank, J., 1998. "Effects of turbulence on species mass fractions in methane/air jet flames". *Symposium*
535 *(International) on Combustion*, **27**(1), pp. 1087–1095.
- 536 [2] Barlow, R., Frank, J., Karpetis, A., and Chen, J.-Y., 2005. "Piloted methane/air jet flames: Transport effects and aspects
537 of scalar structure". *Combustion and Flame*, **143**(4), pp. 433–449.
- 538 [3] Schneider, C., Dreizler, A., Janicka, J., and Hassel, E., 2003. "Flow field measurements of stable and locally extin-
539 guishing hydrocarbon-fuelled jet flames". *Combustion and Flame*, **135**(1), pp. 185–190.

- 540 [4] Lysenko, D., Ertesvåg, I., and Rian, K. E., 2014. “Numerical simulations of the sandia flame d using the eddy dissipa-
541 tion concept”. *Flow Turbulence and Combustion*, **93**, 07.
- 542 [5] Saini, R., Prakash, S., De, A., and Yadav, R., 2018. “Investigation of nox in piloted stabilized methane-air diffusion
543 flames using finite-rate and infinitely-fast chemistry based combustion models”. *Thermal Science and Engineering
544 Progress*, **5**, pp. 144–157.
- 545 [6] Pitsch, H., and Steiner, H., 2000. “Scalar mixing and dissipation rate in large-eddy simulations of non-premixed
546 turbulent combustion”. *Proceedings of the Combustion Institute*, **28**(1), pp. 41–49.
- 547 [7] Pecquery, F., Moureau, V., Lartigue, G., Vervisch, L., and Roux, A., 2014. “Modelling nitrogen oxide emissions in
548 turbulent flames with air dilution: Application to les of a non-premixed jet-flame”. *Combustion and Flame*, **161**(2),
549 pp. 496–509.
- 550 [8] He, D., Yu, Y., Kuang, Y., and Wang, C., 2021. “Model comparisons of flow and chemical kinetic mechanisms for
551 methane–air combustion for engineering applications”. *Applied Sciences*, **11**(9).
- 552 [9] Wronski, T., Schönnenbeck, C., Zouaoui-Mahzoul, N., Brillard, A., and Brilhac, J.-F., 2022. “Numerical simulation
553 through fluent of a cold, confined and swirling airflow in a combustion chamber”. *European Journal of Mechanics -
554 B/Fluids*, **96**, pp. 173–187.
- 555 [10] Launder, B. E. B. E., 1972. *Lectures in mathematical models of turbulence / B. E. Launder and D. B. Spalding*.
556 Academic Press, London.
- 557 [11] Launder, B. E., Reece, G. J., and Rodi, W., 1975. “Progress in the development of a reynolds-stress turbulence closure”.
558 *Journal of Fluid Mechanics*, **68**(3), p. 537–566.
- 559 [12] Wilcox, D. C., et al., 1998. *Turbulence modeling for CFD*, Vol. 2. DCW industries La Canada, CA.
- 560 [13] Meraner, C., Li, T., Ditaranto, M., and Løvås, T., 2018. “Cold flow characteristics of a novel bluff body hydrogen
561 burner”. *International Journal of Hydrogen Energy*, **43**(14), pp. 7155–7168.
- 562 [14] Dutka, M., Ditaranto, M., and Løvås, T., 2016. “Nox emissions and turbulent flow field in a partially premixed bluff
563 body burner with ch4 and h2 fuels”. *International Journal of Hydrogen Energy*, **41**(28), pp. 12397–12410.
- 564 [15] Shih, T.-H., Liou, W. W., Shabbir, A., Yang, Z., and Zhu, J., 1995. “A new $k - \epsilon$ eddy viscosity model for high reynolds
565 number turbulent flows”. *Computers & Fluids*, **24**(3), pp. 227–238.
- 566 [16] Menter, F., Kuntz, M., and Langtry, R., 2003. “Ten years of industrial experience with the sst turbulence model”. *Heat
567 and Mass Transfer*, **4**, 01.
- 568 [17] Menter, F. R., 2012. “Best practice: scale-resolving simulations in ansys cfd”. *ANSYS Germany GmbH*, **1**.
- 569 [18] Menter, F., 2018. “Stress-blended eddy simulation (sbcs)—a new paradigm in hybrid rans-les modeling”. In *Progress
570 in Hybrid RANS-LES Modelling*, Y. Hoarau, S.-H. Peng, D. Schwaborn, and A. Revell, eds., Springer International
571 Publishing, pp. 27–37.
- 572 [19] Sensirion, 2018. Sdp8xx-analog datasheet. https://www.mouser.co.uk/datasheet/2/682/SERI_S_A0004817208_1-2524897.pdf. Accessed on 03 October 2022.
- 574 [20] Ltd., K. E. P. Kimo mp 200 p thermo anemometer manometer datasheet. <https://www.netzerotools.com/>

- 575 assets/images/pdfs/kimo/mp200/NZ-kimo-MP200-manometer-datasheet.pdf. Accessed on 03
576 October 2022.
- 577 [21] Sensirion, 2014. Ta440 thermal anemometer datasheet. [https://tsi.com/getmedia/
578 fd36afb0-67d4-4cf5-a517-cc3743024329/TA410-TA430-440_Thermal_Anemometers\
579 _A4_UK_2980548-web?ext=.pdf](https://tsi.com/getmedia/fd36afb0-67d4-4cf5-a517-cc3743024329/TA410-TA430-440_Thermal_Anemometers_A4_UK_2980548-web?ext=.pdf). Accessed on 03 October 2022.
- 580 [22] Ansys-Inc, 2019. Fluent theory guide, release 2019.r3. [https://www.ansys.com/Products/Fluids/
581 ANSYS-Fluent](https://www.ansys.com/Products/Fluids/ANSYS-Fluent). Accessed on 11 May 2022.
- 582 [23] Morrall, A., Quayle, S., and Campobasso, M., 2020. “Turbulence modelling for rans cfd analyses of multi-nozzle
583 annular jet pump swirling flows”. *International Journal of Heat and Fluid Flow*, **85**, p. 108652.
- 584 [24] Gibson, M. M., and Launder, B. E., 1978. “Ground effects on pressure fluctuations in the atmospheric boundary layer”.
585 *Journal of Fluid Mechanics*, **86**(3), p. 491–511.
- 586 [25] Fu, S., Launder, B., and Leschziner, M., 1987. “Modelling strongly swirling recirculating jet flow with reynolds-stress
587 transport closures”. In 6th Symposium on Turbulent Shear Flows, pp. 17_6_1–17_6_6.
- 588 [26] Launder, B., 1989. “Second-moment closure and its use in modelling turbulent industrial flows”. *International Journal
589 for Numerical Methods in Fluids*, **9**, pp. 963–985.
- 590 [27] Launder, B., 1989. “Second-moment closure: present.. and future?”. *International Journal of Heat and Fluid Flow*,
591 **10**, pp. 282–300.
- 592 [28] Speziale, C. G., Sarkar, S., and Gatski, T. B., 1991. “Modelling the pressure–strain correlation of turbulence: an
593 invariant dynamical systems approach”. *Journal of Fluid Mechanics*, **227**, p. 245–272.
- 594 [29] Menter, F. R., 2009. “Review of the shear-stress transport turbulence model experience from an industrial perspective”.
595 *International journal of computational fluid dynamics*, **23**(4), pp. 305–316.
- 596 [30] Menter, F. R., 1994. “Two-equation eddy-viscosity turbulence models for engineering applications”. *AIAA journal*,
597 **32**(8), pp. 1598–1605.
- 598 [31] von Kármán, T., 1931. “Mechanical similitude and turbulence”.
- 599 [32] Snegirev, A., and Frolov, A., 2011. “The large eddy simulation of a turbulent diffusion flame”. *High Temperature*, **49**,
600 10, pp. 690–703.
- 601 [33] Zhang, Y., Xing, Y., and Chen, S., 2018. “Cfd investigation based on gas burner with low-nox strategy of fuel-staging”.
602 *IOP Conference Series: Earth and Environmental Science*, **153**(3), may, p. 032022.
- 603 [34] Du, H., Li, Z., Liu, Z., Zhang, M., Chen, Z., Song, J., Fang, F., and Xiao, R., 2022. “Influence of the parallel c-layer
604 secondary air on flow, combustion and nox generation characteristics of a 600mwe fw down-fired boiler retrofitted with
605 a stable combustion organization mode”. *Combustion Science and Technology*, **0**(0), pp. 1–25.

606 **List of Tables**

607 1 Dimensions of primary inlet duct, gas burner, and combustion chamber normalized by nozzle external diam-
608 eter d_n 23

609 2 Mass flow rates and velocity RMS^A values on cross sections at positions L1, L2 and E3 computed with
610 coarse, medium and fine grid $k - \omega$ SST P-WF set-up for operating condition V90. 24

611 3 Air mass flow rate estimated experimentally from the measured velocities and computed by the CFD pre-
612 scribing the inlet total pressure for the V90 and V50 operating conditions 25

613 4 Reference pressures measured and computed by the CFD prescribing the inlet velocities for the V90 and
614 V50 operating conditions. 26

615 5 Nondimensionalized static pressure p^* in the fuel inlet box evaluated with different set-ups for V90 and V50
616 conditions 27

617 6 Nondimensionalized static pressure p^* in the fuel inlet box measured and evaluated with different turbulence
618 models for V90 and V50 conditions 28

619 **List of Figures**

620 1 Studied gas burner. 29

621 2 Inner views of gas burner. 30

622 3 Geometry parameters of primary inlet duct and gas burner. 31

623 4 Schematic of station 1. 32

624 5 Schematic of station 2. Black dots indicate pressure tap positions. 33

625 6 Schematic of station 3, and valve regulation. 34

626 7 Main dimensions and outer boundaries of physical domain. 35

627 8 Longitudinal section of medium refinement grid for analysis of V90 operating condition resolving boundary
628 layers down to wall. 36

629 9 Measurement-based data for primary inlet boundary conditions at V90 and V50 conditions. 37

630 10 Profiles of V_x velocity components on transversal lines L1, L2 and E3 computed with coarse, medium and
631 fine grid $k - \omega$ SST P-WF set-up for operating condition V90. Middle subplot also reports profiles of V_r
632 velocity component. 38

633 11 Wall pressure profiles on lines F1 computed with coarse, medium and fine grid $k - \omega$ SST P-WF set-up for
634 operating condition V90. 39

635 12 Analysis of solution sensitivity to inlet boundary conditions and wall BL modeling: computed and measured
636 V_x profiles on transversal lines E1, E2, and E3 in combustion chamber. 40

637 13 Analysis of solution sensitivity to inlet boundary conditions and wall BL modeling: computed and measured
638 nondimensionalized static pressure at V90 condition. 41

639 14 Analysis of solution sensitivity to turbulence model: computed and measured V_x profiles on transversal lines
640 E1, E2, and E3 in combustion chamber. 42

641 15 Velocity field in burner case at V90 condition computed with RSM-BSL P-WR. 43

642 16 Velocity field in downstream part of burner case at V90 condition computed with RSM-BSL P-WR. 44

643 17 Nozzle velocity field at V90 condition computed with RSM-BSL P-WR. 45

644 18 Nondimensionalized static pressure field in burner case at V90 condition computed with RSM-BSL P-WR. 46

645 19 Measured and RSM-BSL P-WR profiles of nondimensionalized static pressure at V90 condition. 47

646 20 Measured and RSM-BSL P-WR V_x profiles on transversal lines E1, E2, and E3 in combustion chamber at
647 condition V90. 48

648 21 Velocity field in burner case at V50 condition computed with RSM-BSL P-WR. 49

649 22 Velocity field in downstream part of burner case at V50 condition computed with RSM-BSL P-WR. 50

650 23 Nozzle velocity field at V50 condition computed with RSM-BSL P-WR. 51

651 24 Measured and RSM-BSL P-WR profiles of nondimensionalized static pressure at V50 condition. 52

652 25 Measured and RSM-BSL P-WR V_x profiles on transversal lines E1, E2 and E3 in combustion chamber at
653 condition V50. 53

Table 1: Dimensions of primary inlet duct, gas burner, and combustion chamber normalized by nozzle external diameter d_n .

l_n/d_n	l_b/d_n	h_b/d_n	l_{in}/d_n	l_{x_b}/d_n	l_{x_n}/d_n	l_c/d_n	d_c/d_n
1.00	2.48	1.36	6.40	0.66	0.80	16.00	6.40

Table 2: Mass flow rates and velocity RMS^A values on cross sections at positions L1, L2 and E3 computed with coarse, medium and fine grid $k - \omega$ SST P-WF set-up for operating condition V90.

Load Condition	$\dot{m}_{air} [\frac{kg}{s}]$	$RMS^A_{L1} [\frac{m}{s}]$	$RMS^A_{L2} [\frac{m}{s}]$	$RMS^A_{E3} [\frac{m}{s}]$
Coarse	$6.310 \cdot 10^{-2}$	2.971	1.022	0.895
Medium	$6.334 \cdot 10^{-2}$	3.210	1.043	0.980
Fine	$6.333 \cdot 10^{-2}$	3.269	1.052	0.993

Table 3: Air mass flow rate estimated experimentally from the measured velocities and computed by the CFD prescribing the inlet total pressure for the V90 and V50 operating conditions

Load Condition	$\dot{m}_{air} [\frac{Kg}{s}]$			$\Delta_{CFD} [\%]$
	<i>Exp_{est}</i>	P-WR	P-WF	
V90	$6.95 \cdot 10^{-2}$	$5.97 \cdot 10^{-2}$	$6.33 \cdot 10^{-2}$	6.0
V50	$5.79 \cdot 10^{-2}$	$5.33 \cdot 10^{-2}$	$5.62 \cdot 10^{-2}$	5.4

Table 4: Reference pressures measured and computed by the CFD prescribing the inlet velocities for the V90 and V50 operating conditions.

Load Condition	$p_{r0}[Pa]$		
	Exp-SDP816	V-WR	V-WF
V90	356.5	486.7	444.7
V50	424.7	515.5	464.5

Table 5: Nondimensionalized static pressure p^* in the fuel inlet box evaluated with different set-ups for V90 and V50 conditions

Load Condition	$p_{FB}^*[-]$				
	Exp-SDP816	V-WR	V-WF	P-WR	P-WF
V90	-0.107	-0.111	-0.121	-0.111	-0.120
V50	-0.062	-0.070	-0.077	-0.072	-0.080

Table 6: Nondimensionalized static pressure p^* in the fuel inlet box measured and evaluated with different turbulence models for V90 and V50 conditions

Load Condition	$p_{FB}^*[-]$			
	Exp-SDP816	$k-\omega$ SST	RSM-BSL	RSM- ϵ
V90	-0.107	-0.111	-0.102	-0.082
V50	-0.062	-0.072	-0.069	-0.059

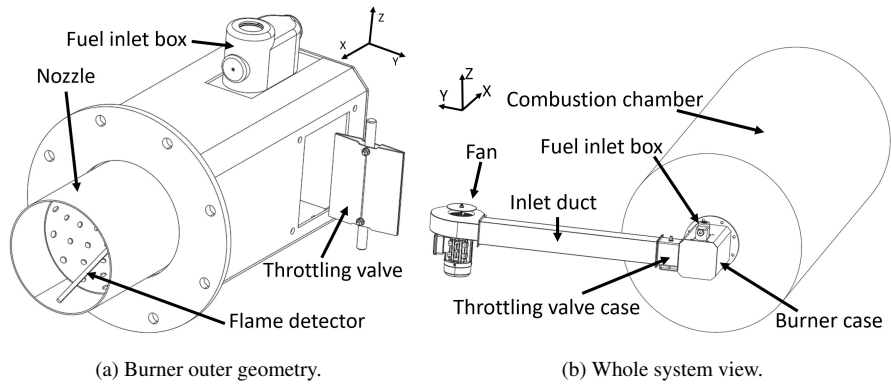


Fig. 1: Studied gas burner.

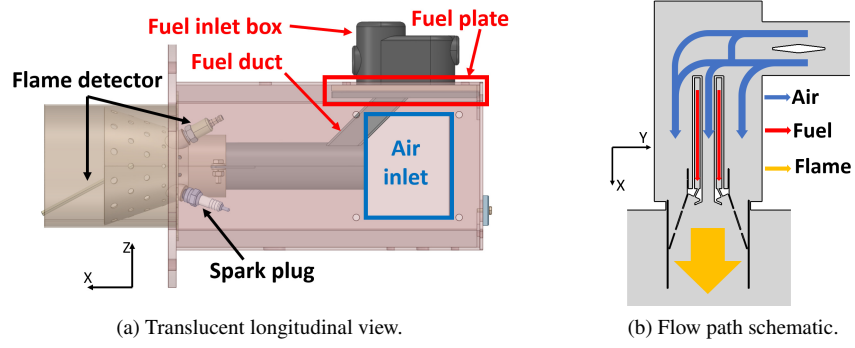


Fig. 2: Inner views of gas burner.

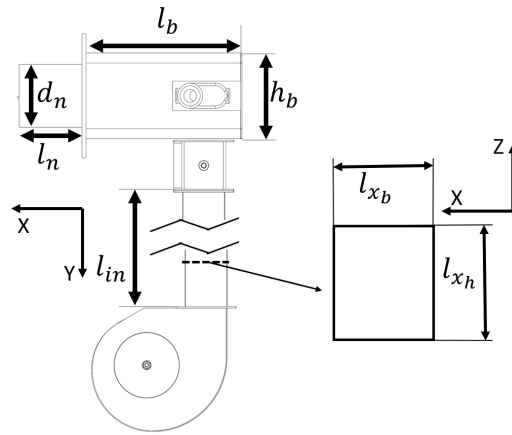


Fig. 3: Geometry parameters of primary inlet duct and gas burner.

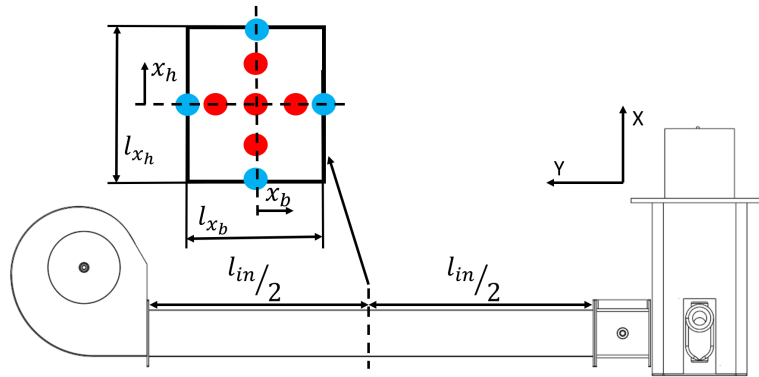


Fig. 4: Schematic of station 1.

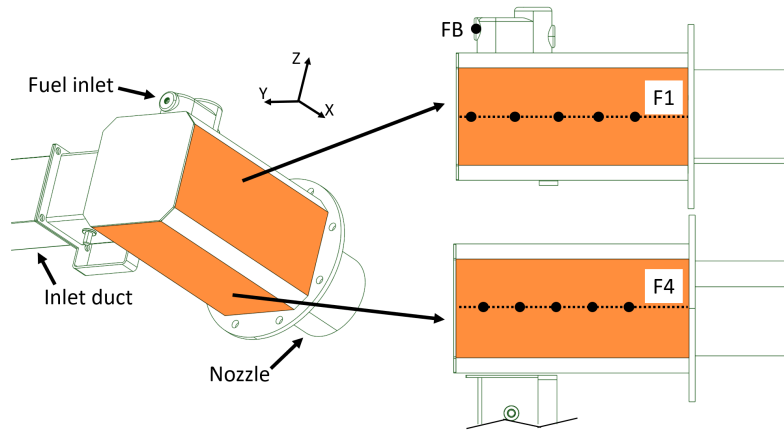
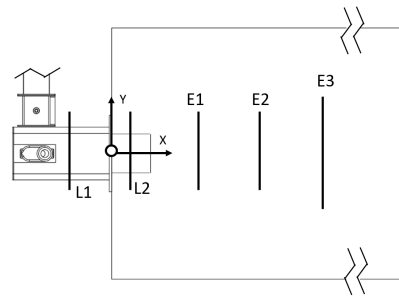
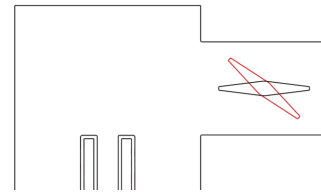


Fig. 5: Schematic of station 2. Black dots indicate pressure tap positions.



(a) Longitudinal positions of velocity traverses.



(b) Valve position at full load operation V90 (black), and partial load operation V50 (red).

Fig. 6: Schematic of station 3, and valve regulation.

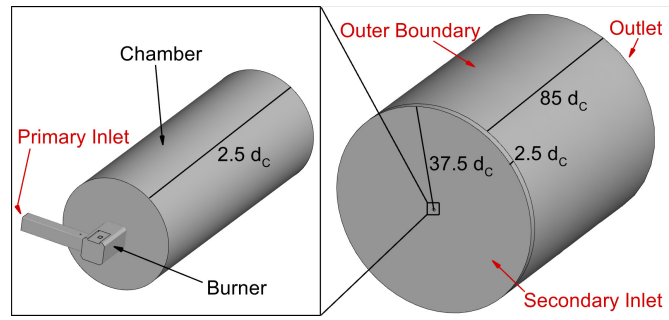


Fig. 7: Main dimensions and outer boundaries of physical domain.

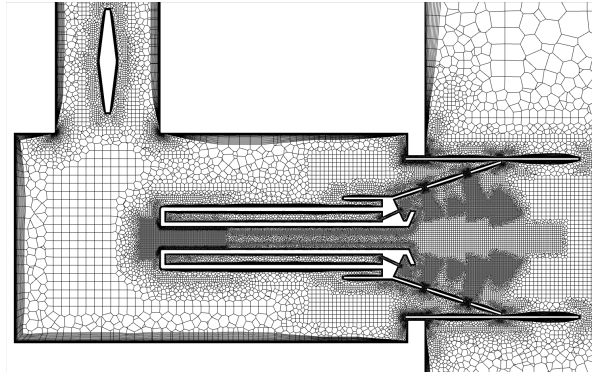
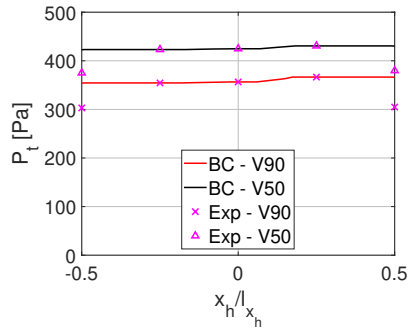
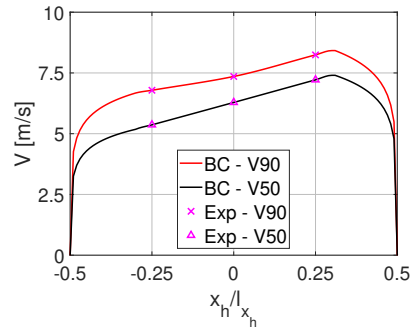


Fig. 8: Longitudinal section of medium refinement grid for analysis of V90 operating condition resolving boundary layers down to wall.



(a) Total pressure profiles.



(b) Velocity profiles.

Fig. 9: Measurement-based data for primary inlet boundary conditions at V90 and V50 conditions.

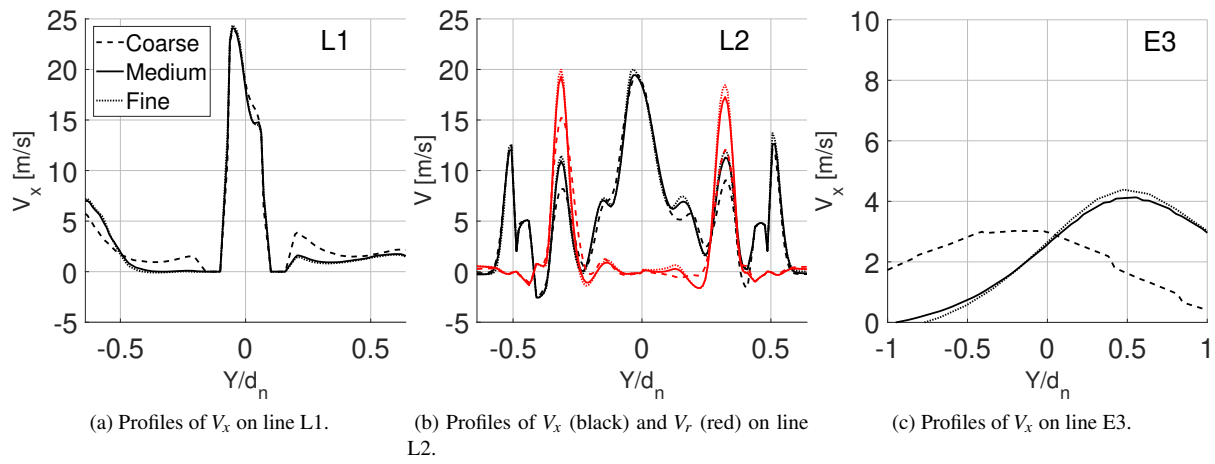


Fig. 10: Profiles of V_x velocity components on transversal lines L1, L2 and E3 computed with coarse, medium and fine grid $k - \omega$ SST P-WF set-up for operating condition V90. Middle subplot also reports profiles of V_r velocity component.

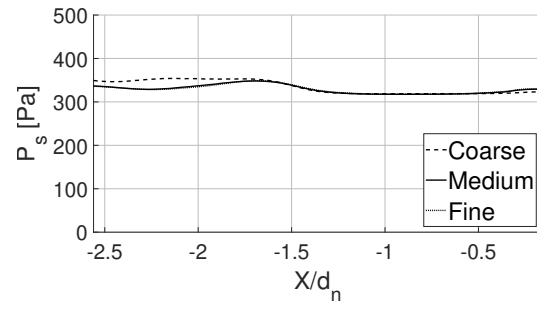
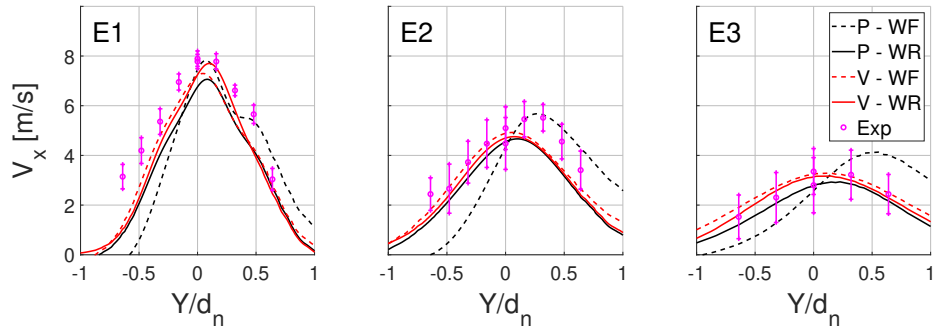
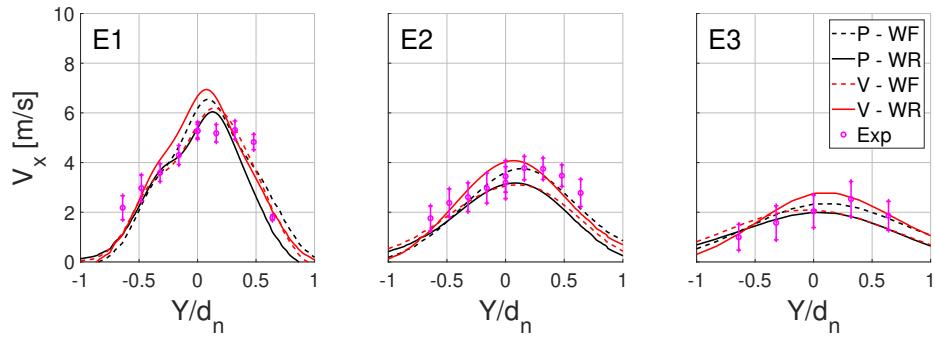


Fig. 11: Wall pressure profiles on lines F1 computed with coarse, medium and fine grid $k - \omega$ SST P-WF set-up for operating condition V90.



(a) V90 operating condition.



(b) V50 operating condition.

Fig. 12: Analysis of solution sensitivity to inlet boundary conditions and wall BL modeling: computed and measured V_x profiles on transversal lines E1, E2, and E3 in combustion chamber.

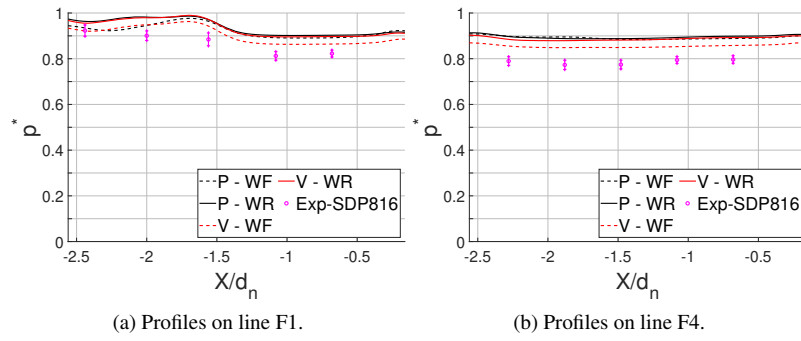


Fig. 13: Analysis of solution sensitivity to inlet boundary conditions and wall BL modeling: computed and measured nondimensionalized static pressure at V90 condition.

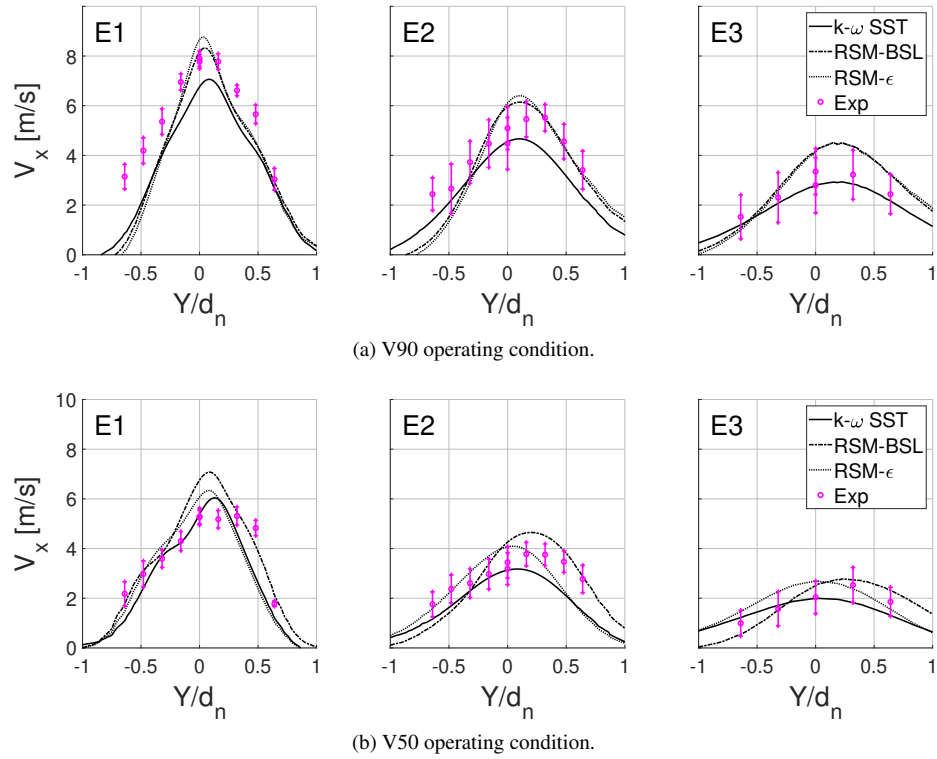


Fig. 14: Analysis of solution sensitivity to turbulence model: computed and measured V_x profiles on transversal lines E1, E2, and E3 in combustion chamber.

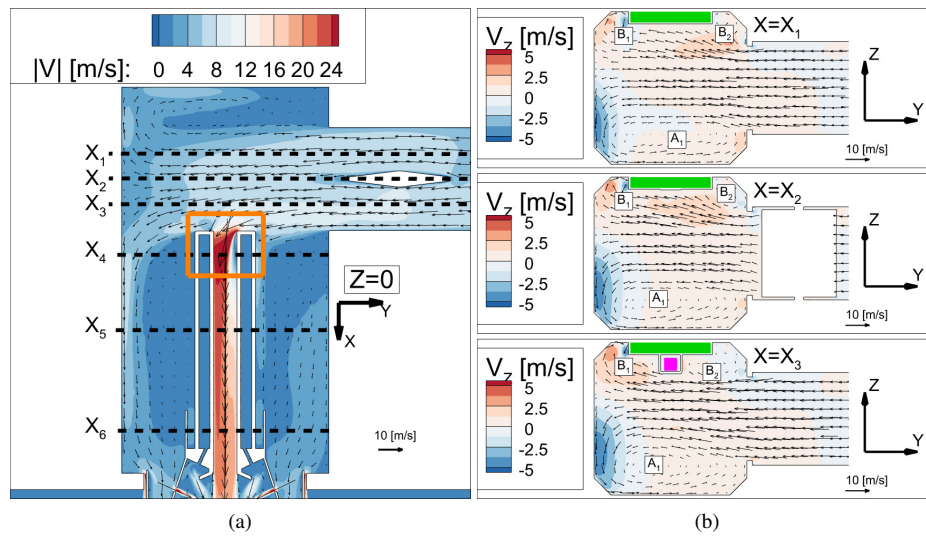


Fig. 15: Velocity field in burner case at V90 condition computed with RSM-BSL P-WR.

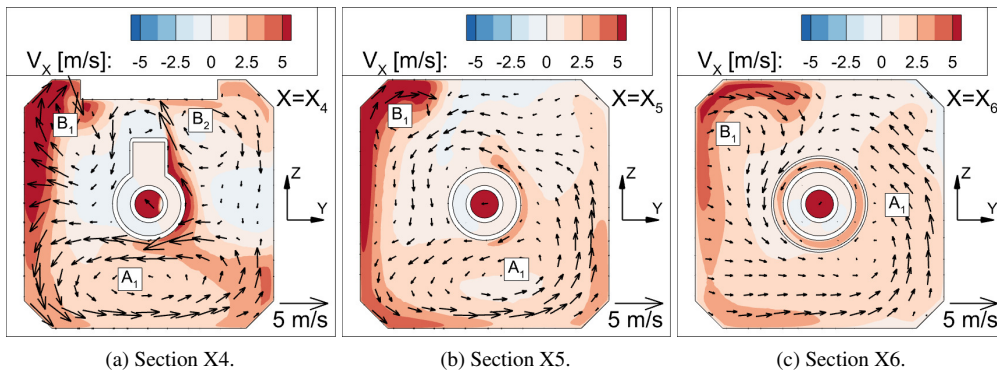


Fig. 16: Velocity field in downstream part of burner case at V90 condition computed with RSM-BSL P-WR.

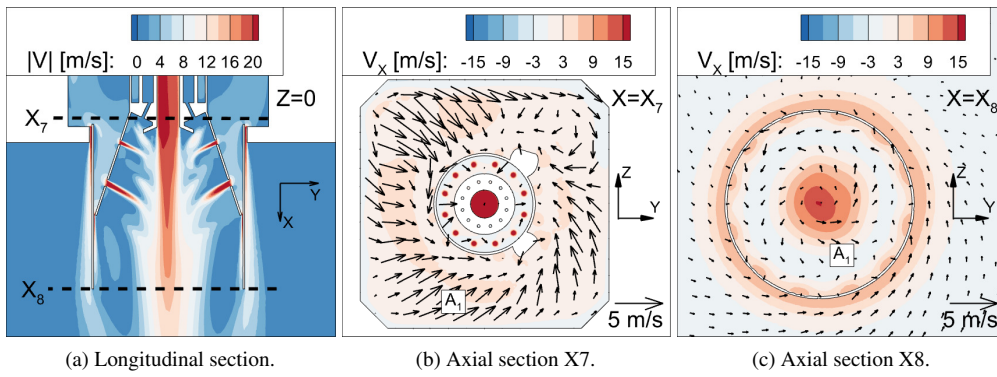


Fig. 17: Nozzle velocity field at V90 condition computed with RSM-BSL P-WR.

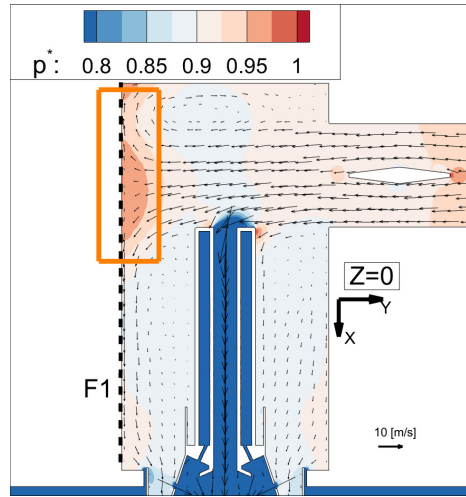


Fig. 18: Nondimensionalized static pressure field in burner case at V90 condition computed with RSM-BSL P-WR.

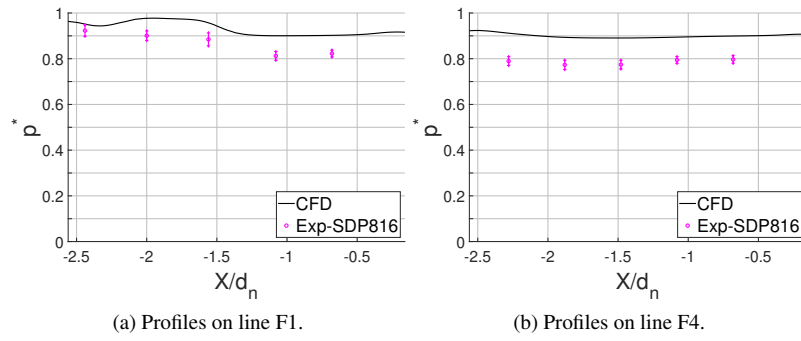


Fig. 19: Measured and RSM-BSL P-WR profiles of nondimensionalized static pressure at V90 condition.

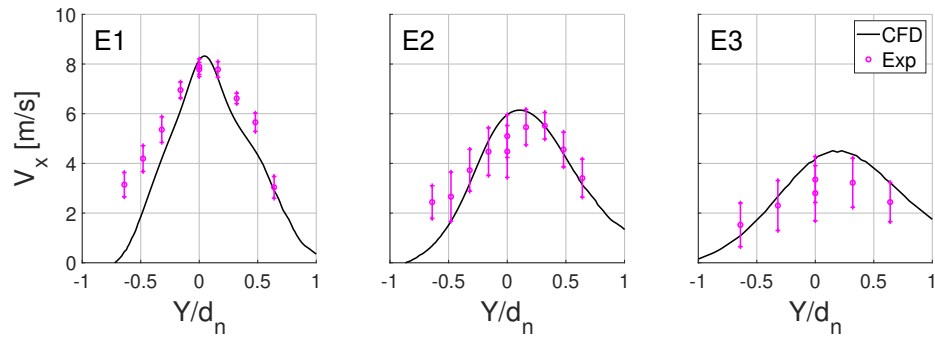


Fig. 20: Measured and RSM-BSL P-WR V_x profiles on transversal lines E1, E2, and E3 in combustion chamber at condition V90.

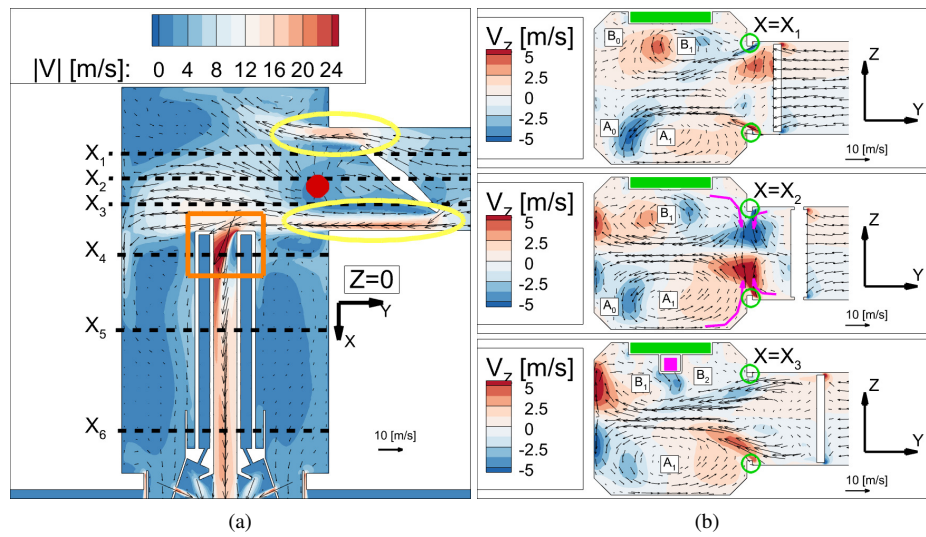


Fig. 21: Velocity field in burner case at V50 condition computed with RSM-BSL P-WR.

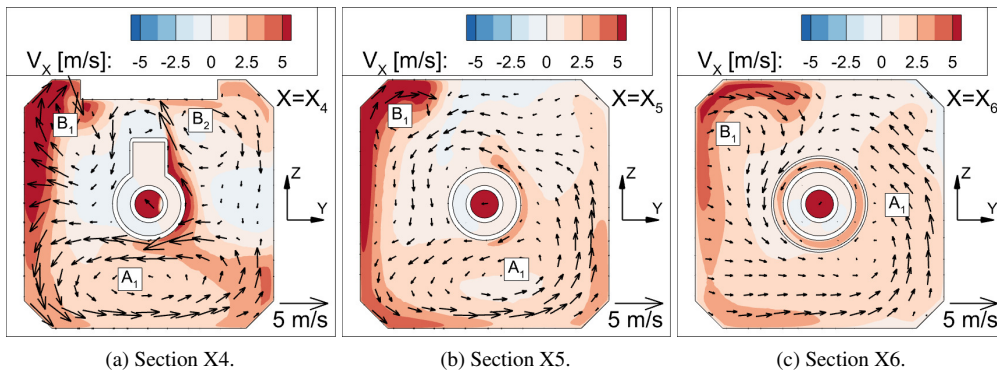


Fig. 22: Velocity field in downstream part of burner case at V50 condition computed with RSM-BSL P-WR.

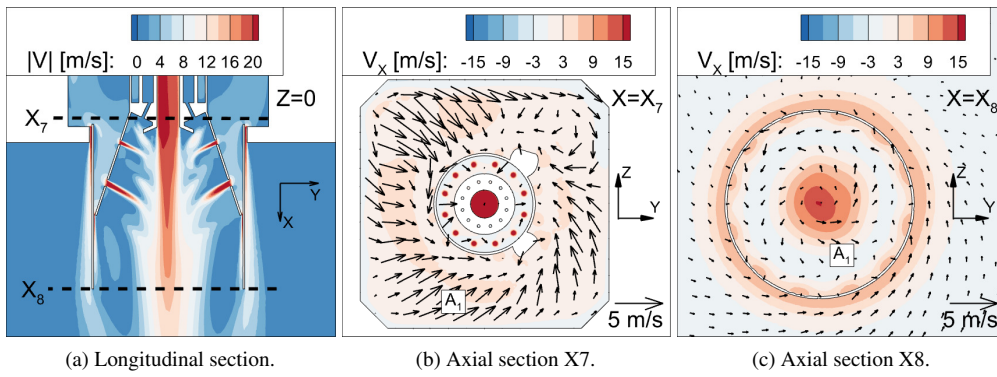


Fig. 23: Nozzle velocity field at V50 condition computed with RSM-BSL P-WR.

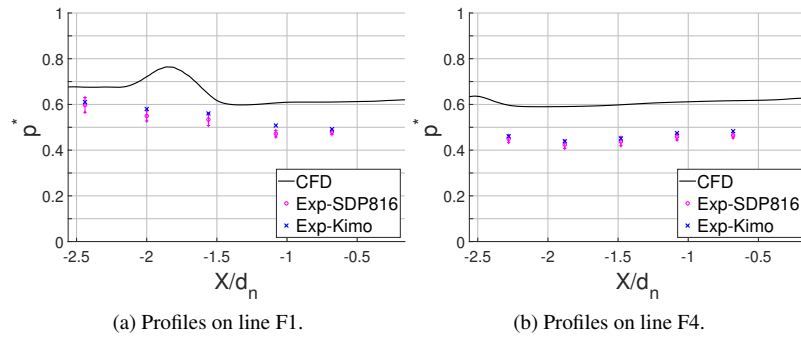


Fig. 24: Measured and RSM-BSL P-WR profiles of nondimensionalized static pressure at V50 condition.

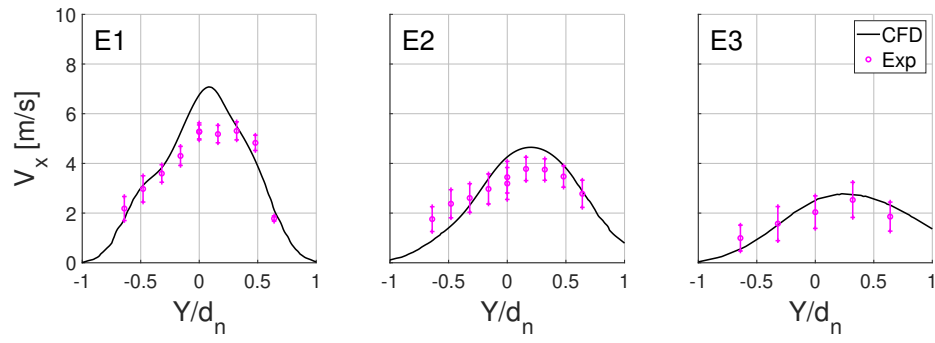


Fig. 25: Measured and RSM-BSL P-WR V_x profiles on transversal lines E1, E2 and E3 in combustion chamber at condition V50.