



HAL
open science

Advanced benchmark of the flow through a mixing vane grid - Large Eddy Simulation validation

Benjamin Farges, Marie-Charlotte Gauffre, Sofiane Benhamadouche, Pierre Badel, Vincent Faucher, Guillaume Ricciardi

► **To cite this version:**

Benjamin Farges, Marie-Charlotte Gauffre, Sofiane Benhamadouche, Pierre Badel, Vincent Faucher, et al.. Advanced benchmark of the flow through a mixing vane grid - Large Eddy Simulation validation. Nuclear Engineering and Design, 2021, 381, pp.111335. 10.1016/j.nucengdes.2021.111335 . cea-03409431

HAL Id: cea-03409431

<https://cea.hal.science/cea-03409431>

Submitted on 29 Oct 2021

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.



Distributed under a Creative Commons Attribution - NonCommercial - NoDerivatives 4.0 International License

Advanced benchmark of the flow through a mixing vane grid – Large Eddy Simulation validation

Farges Benjamin

Framatome
10 rue Juliette Récamier, Lyon, FR-69006
benjamin.farges@framatome.com

Gauffre Marie-Charlotte

EDF
19 Rue Pierre Bourdeix, 69007 Lyon
marie-charlotte.gauffre@edf.fr

Benhamadouche Sofiane

EDF Lab
6 quai Watier, Chatou, FR-78400
sofiane.benhamadouche@edf.fr

Badel Pierre

EDF Lab
Boulevard Gaspard Monge, 91120 Palaiseau
Pierre-bernard.badel@edf.fr

Faucher Vincent*

CEA, DES, IRESNE, Nuclear Technology Department,
Center of Cadarache, Saint-Paul-lez-Durance, FR-13108
vincent.faucher@cea.fr

Ricciardi Guillaume

CEA, DES, IRESNE, Nuclear Technology Department,
Center of Cadarache, Saint-Paul-lez-Durance, FR-13108
guillaume.ricciardi@cea.fr

1 **ABSTRACT**

2
3 Identifying what causes fuel assembly vibrations downstream of Mixing Vane Grids (MVG) in Pressurized
4 Water reactor (PWR) is of paramount importance for nuclear community to understand grid-to-rod fretting
5 wear. Experiments, called CALIFS, were carried out by the Atomic Energy Commission (CEA) on a 5x5
6 MVG at a hydraulic Reynolds number of 66,000, in order to measure the flow velocity and the pressure
7 along the central rod. In parallel, a benchmark for Large Eddy Simulation (LES) was setup to compare the
8 predictions of three different CFD codes: Star-CCM+, *Code_Saturne* and TrioCFD to experimental
9 measurements. The computational domain is representative of a span of CALIFS mockup, composed of a
10 5x5 rod bundle with a MVG. The three computations overall give very satisfactory results, independently
11 from the mesh created and the modelling options selected. It seems to suggest that whatever the software
12 used, this kind of calculations has reached a significant level of robustness and accuracy. Nonetheless, some
13 discrepancies remain concerning the predictions of pressure standard deviation decay far downstream of
14 the mixing vane grid.

16 **KEYWORDS**

17 Pressurized Water Reactor fuel assembly, 5x5 mixing vane grid, CALIFS experiment, CFD benchmark,
18 Large Eddy Simulation, pressure standard deviation.

21 **1 INTRODUCTION: INDUSTRIAL CHALLENGE AND PURPOSE OF THE PAPER**

22 Flow-induced vibrations (FIV) are a major research topic for nuclear reactor technology in general, and for
23 the Pressurized Water Reactors in particular. Beyond the need to always prevent fluidelastic instability
24 phenomena which could lead to rapid failures, it is also necessary to account for the long-time effects of
25 the flow turbulence excitations. For instance, the grid-to-rod fretting wear is still a worldwide dominant
26 fuel rod leaker mechanism [1] and its main cause has been identified as fuel rod vibration induced by the
27 turbulent flow [2].

28 This topic is currently tackled through dedicated experiments. Carrying out the associated simulation is still
29 a challenging field for *Computational Fluid Dynamics* (CFD) software due the stochastic and turbulence-
30 induced nature of the fluid loading applied to the structures at rod/grid level within a fuel assembly.
31 Benhamadouche et al. [11] carried out wall-modelled LES of the flow through a 4x4 mixing vane grid at a
32 Reynolds number of 40,000, in order to compute the pressure load. This latter has been used combined to
33 a beam equation to predict the displacements obtained along a rod and the qualitative results corresponded

34 to what is observed in reality (few microns). However, this computation cannot be considered as validated
35 and the methodology has to be confronted to experimental data. A dedicated Round Robin exercise to the
36 flow through Simple Support Grids and Mixing Vane Grids has been carried out by EPRI [12]. This
37 benchmark included heat transfer. Almost only RANS models have been employed in this benchmark and
38 the ability to capture pressure fluctuations has not been addressed.

39 Many recent attempts to simulate the fluid flow in a fuel assembly can be found in the literature. Bieder et
40 al. [16] showed that, downstream close to the mixing vane the turbulence is isotropic and anisotropic further
41 downstream. Chen et al. in [17] showed that simulation should count at least a 4 by 4 rod bundle to represent
42 a full bundle. The turbulent intensity generated by the mixing vanes is underestimated by standard *k-epsilon*
43 simulations [18], LES simulation perform better especially far from the grid [19]. The design of mixing van
44 has as strong influence on the turbulence [20], more specifically the inclination of the vanes increases the
45 turbulent intensity [21]. Spring and dimple have also an influence on the flow [22]. The MATiS-H
46 benchmark conducted by KAERI showed that simulation accounting for LES model with the finest mesh
47 does not guarantee the best results [23].

48 The purpose of the present article is to contribute to the definition of the state-of-the-art for industrial CFD
49 computation of single phase highly turbulent flow in rod bundles involving mixing grids. A particular focus
50 is dedicated to pressure fluctuations. The work involves cross-comparisons of predictions provided by three
51 different CFD solvers with various user-environment and objectives as well as various implemented
52 numerical methods and models for the case of interest, all evaluated with respect to a reference experiment
53 named CALIFS (see for instance [3] and [4]) described further in this introduction.

54 Such a benchmark, with results produced mobilizing a significantly high level of expert knowledge in the
55 use of each of considered software, is able to provide relevant insights regarding the capabilities of turbulent
56 CFD simulation going far beyond the proposed application and contribute to the definition of design
57 processes increasingly involving numerical results, with mandatory uncertainty assessment obtained from
58 detailed calculation at local scale.

59

60 **2 DESCRIPTION OF THE CALIFS 5X5 REFERENCE EXPERIMENT**

61 CALIFS 5x5, illustrated in Figure 1, is a water rig working within the ranges 0-400 m³/h, 10°-70°C, 0-10
62 bars. The length of the test section is about 2.5 m. The flow cross section is a square of 184mm side. A 5x5
63 rods bundle, using rods of diameter 26.9 mm, is placed within the test section using spacer-grids, with one
64 grid of interest with full optical access (see Figure 1-c and Figure 1-d). The hydraulic diameter (denoted D_h)

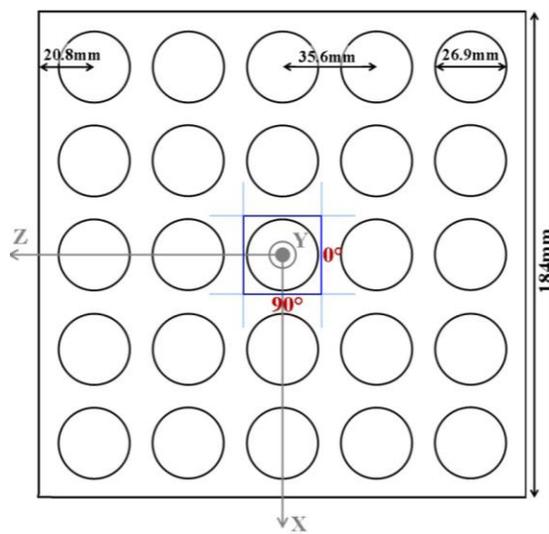
65 in the following) of the bundle is 27.7 mm and is used as the reference length scale. The scale of the
66 experiment is higher than 1 in order to use appropriate sensors to measure the pressure along the rods.

67 The grids are made with 1.2mm thick plates of stainless steel 304L. The height of the plates is 93 mm and
68 they are assembled perpendicularly to design a mesh with 25 cells. Within each cell, the rods are sustained
69 radially using dimples and springs-blades, representative of a realistic Mixing Vane Grid (MVG). The
70 dimples are made of TEFLON¹ and the springs-blades are made with 1 mm thick blades of stainless steel
71 301 T4.

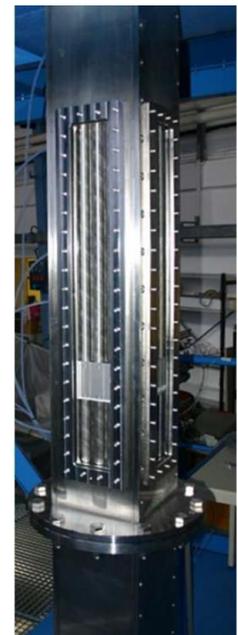
72



(a) General view of CALIFS 5x5 test rig

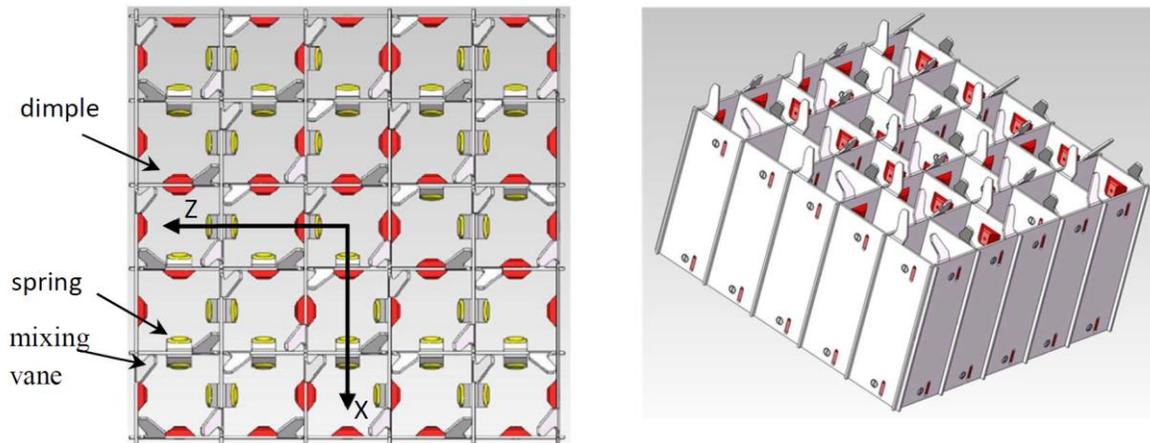


(b) Main dimensions and orientations



(c) View of the test section

¹ TEFLON is a registered trademark of E. I. du Pont de Nemours and Company or its affiliates.



(d) View of the details of the mixing grid equipped with vanes

73

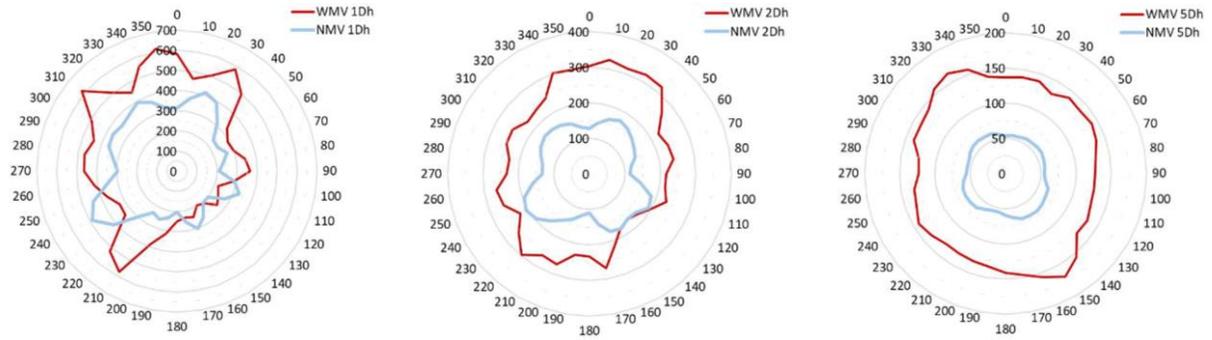
Figure 1. Some details about the CALIFS 5x5 reference experiment

74 The tests are performed for a flow rate velocity of $2.4 \text{ m}\cdot\text{s}^{-1}$ and a flow temperature of 20°C , yielding a
 75 reference hydraulic Reynolds number of 66,000, enough to significantly challenge the simulation results in
 76 the turbulent regime.

77 Global pressure drops along the bundle and unsteady local pressure measurements are implemented, the
 78 latter being obtained by instrumenting the central rod of the 5x5 rods bundle with a high-resolution piezo-
 79 resistive sensor. The pressure distribution around the central rod circumference is obtained by turning the
 80 instrumented rod with an increment of 10° over 360° . This azimuthal distribution is measured at various
 81 longitudinal positions downstream of the grid of interest by translating the instrumented rod. The
 82 longitudinal distances which are studied are $0.5D_h$, $1D_h$, $2D_h$, $3D_h$, $4D_h$, $5D_h$, $10D_h$, $15D_h$ and $20D_h$
 83 downstream of the mixing grid, respectively. Figure 2 shows an example extracted from [4] of the
 84 experimental distribution of pressure standard deviation against which numerical results will be evaluated.
 85 Only the red curves are considered in the present paper, the blue curves being obtained with a preliminary
 86 configuration without mixing vanes.

87 Average velocity profiles in the tube bundle are also provided through Laser Doppler Velocimetry (LDV)
 88 to serve also as experimental references for numerical results evaluation.

89



(a) $1D_h$ downstream of the MVG (b) $2D_h$ downstream of the MVG (c) $5D_h$ downstream of the MVG

90 **Figure 2. Pressure standard deviation obtained in CALIFS 5x5 experiment at different distances**
 91 **from the mixing vane grid along the central rod (only the red curves have to be considered, the blue**
 92 **curves were obtained with a preliminary configuration without mixing vanes)**

93
 94

95 **3 NUMERICAL MODELS**

96 Since the purpose of the benchmark was to challenge and compare complete numerical simulations
 97 strategies for highly turbulent flow in a bundle configuration, numerical models mimicking CALIFS 5x5
 98 experiment were built separately by Framatome, EDF and CEA using their usual solvers with their state of
 99 the art of expertise. The selected CFD programs are given in Table 1.

100

Framatome	<p>Star-CCM+ V11.2</p> <p>(https://www.plm.automation.siemens.com/global/fr/products/simcenter/Star-CCM.html)</p> <p>A fully industrial software solution from SIEMENS for design and engineering, including advanced CFD and co-simulation. ²</p>
EDF	<p><i>Code_Saturne</i> V5.0 (https://www.code-saturne.org/)</p> <p>A highly customizable open-source software for single phase laminar or turbulent flows developed by EDF, for many kinds of applications in the field of energy including in-core hydraulics for nuclear, external atmospheric flows and flows interacting with rotors within steam turbine.</p>
CEA	<p>TrioCFD (http://trio CFD.cea.fr)</p> <p>A multi-purpose research open-source solution developed by CEA to provide high-resolution solutions for complex flows with interfaces, laminar or turbulent and designed for advanced multiphysical applications and couplings.</p>

Table 1. Selected software for Framatome, EDF and CEA

101

102

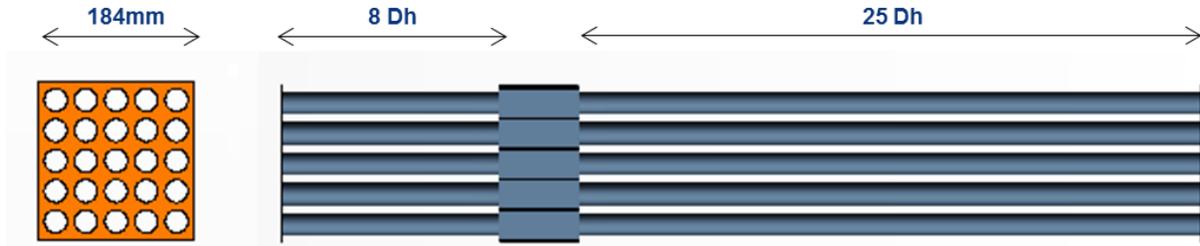
103 The use of Large Eddy Simulation (see for instance [5] or [11]) for turbulence modelling was however
 104 imposed for all contributions, following the results obtained from previous work on a simplified yet
 105 representative configuration with one single rod inside one grid cell placed in a turbulent annular flow (see
 106 [6]).

107 **3.1 Computational domain and boundary conditions**

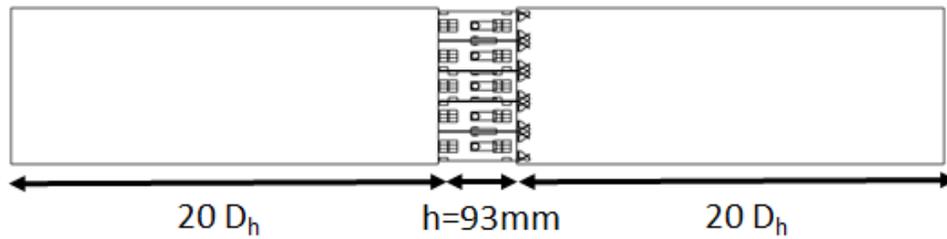
108 The characteristics of the computational domains considered by Framatome, EDF and CEA respectively
 109 are given in Figure 3.

110

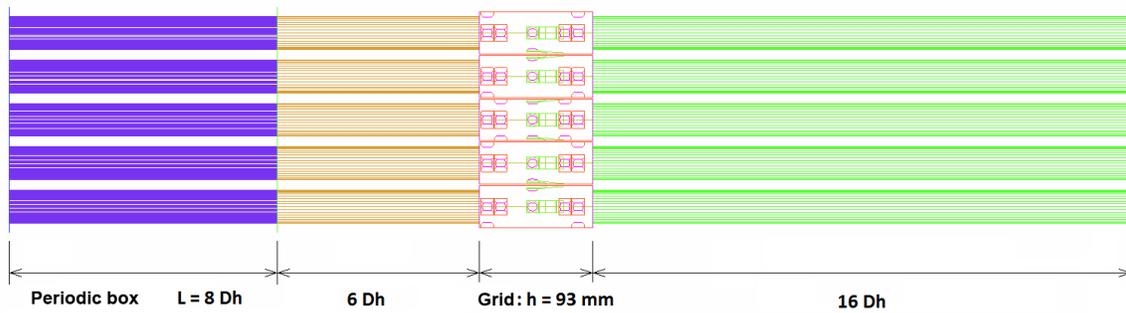
² Star-CCM+® and any and all SIEMENS brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of SIEMENS in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners



111
 112 (a) Framatome computational domain, extending from $8D_h$ upstream the mixing grid of interest to $25D_h$ downstream,
 113 periodic boundary conditions are applied



114
 115 (b) EDF computational domain, extending from $20D_h$ upstream the mixing grid of interest to $20D_h$ downstream,
 116 implicit periodic boundary conditions are applied



118
 119 (c) CEA computational domain, extending from $6D_h$ upstream the mixing grid of interest to $16D_h$ downstream, and
 120 implementing a specific periodic box for turbulent inlet

121 **Figure 3. Computational domains for Framatome, EDF and CEA**

122
 123 For Framatome, the periodicity is achieved by mapping of the flow field (pressure and velocity) between
 124 inlet and outlet to simulate periodic conditions. This mapping operation consists in extracting a set of data
 125 from a given boundary surface (velocity or pressure field, for instance) in order to set it to another boundary.
 126 This yields a three-phase calculation sequence:

- 127 - Phase 1: without any periodic condition, a constant velocity profile is set at the inlet and a constant
 128 pressure profile is set at the outlet. This phase lasts 5 flow passes ;

- 129 - Phase 2a: the mapped velocity is set at the inlet but the pressure boundary conditions remains
130 unchanged. This phase lasts 5 flow passes;
- 131 - Phase 2b: the mapped pressure profile is set at the outlet. Both periodicity conditions are set. This
132 phase lasts 5 flow passes;
- 133 - Phase 3: after solution stabilization of the first phases, data is collected after 15 flow passes.

134 For EDF, periodic conditions of translation are used in the stream-wise direction, with an imposed pressure
135 gradient calculated from the flow rate.

136 CEA suggests a different approach with the initialization of the turbulent velocity field at the bottom entry
137 of the domain ($6D_h$ upstream of the grid) obtained through a so-called periodic recirculation box (beginning
138 $8D_h$ upstream of the bottom entry), where the bundle turbulence is fully established and from which the
139 velocity conditions are extracted and imposed at each time step at the inlet of the physical domain. Pressure
140 is imposed at the domain outlet.

141 In order to perform industrial and affordable computations, wall-modeled LES is used, especially by EDF
142 and CEA (see below for the particular case of Framatome). This approach might give correct mean values
143 (see Benhamadouche [13]). However, its use is still subject to discussion and objections (see Piomelli [14])
144 and this approach using wall functions concepts can also be seen as a very first step of hybrid RANS/LES
145 technique (using a logarithmic profile for example). Table 2 describes the boundary conditions considered
146 along the walls.

Framatome	EDF, CEA
<p>So-called <i>all-y^+</i> approach from Star-CCM+, where the viscous sublayer is fully resolved for low y^+ (a standard wall law is applied for high y^+) This approach benefits from a specifically refined mesh of the boundary layer along the internal walls (see Paragraph 2.2 for meshing processes).</p>	<p>A classical logarithmic wall function is used to predict the wall shear stress and is active on each wall computational cell :</p> $\begin{cases} \frac{U_I}{u^*} = \frac{1}{\kappa} \ln(y^+) + E & \text{if } y^+ > 11.8 \\ \frac{U_I}{u^*} = y^+ & \text{otherwise} \end{cases}$ <p>with U_I the tangential velocity, $y^+ = \frac{\rho y u^*}{\mu}$ the non-dimensional distance to the wall, ρ the density and u^* the friction velocity.</p> <p>Solving for u^* yields the shear stress $\rho(u^*)^2$.</p>

147
148

Table 2. Wall boundary conditions

149 3.2 Meshing processes

150 The mesh used by Framatome for this simulation includes the following features.

- 151 - The cells used for meshing are trimmed, hexahedral cells, with local refinements especially around
152 the grid surface.
- 153 - A prism layer mesher was used in order to resolve the turbulent boundary layer at the rod surface,
154 two layers of prism cells were set on the wall surfaces. This prism layer thickness corresponds to
155 the linear viscous sublayer, calculated thanks to the Dean correlation.
- 156 - Then, the mesh was extruded upwards and downwards of the grid to fully represent the bundle part
157 of the geometry. **The mesh is conformal between inlet and outlet faces for an immediate mapping
158 of the flow field (pressure and velocity) to simulate periodic conditions at stated in Section 3.1.**

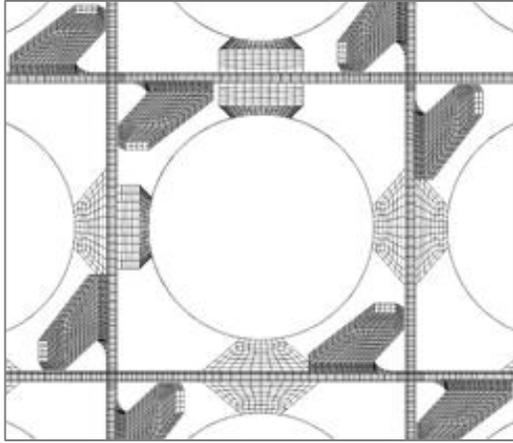
159 The resulting mesh is composed of 35 million fluid cells.

160 Concerning EDF, the computational mesh (created with ICEM CFD v15.0, see Figure 4 (a) and (b) for some
161 details) is composed of 42 million hexahedra and fully conformal. Even if such a mesh is difficult to create
162 and time consuming, these mesh properties are extremely important, to not introduce numerical diffusion,
163 which is not suitable for LES. It was verified *a posteriori* that the logarithmic wall function is active almost
164 everywhere, except in some locations in the grid which complied with the wall-modeled LES carried out
165 (globally $y^+ > 20$). The periodic top and bottom computational faces are also fully conformal. For more
166 details about the numerics and the results, see [15].

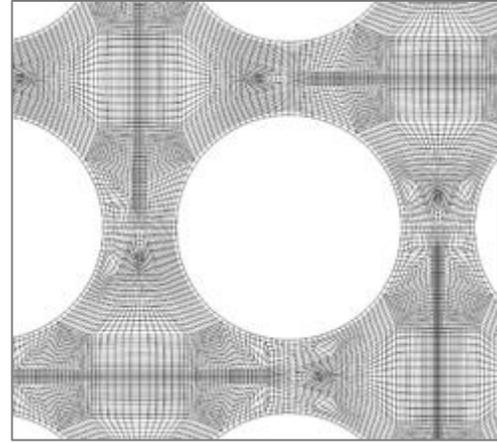
167 Finally, in the version used by CEA for the present work, TrioCFD requires a tetrahedral mesh, which has
168 the benefit of being easily generated in complex geometries such as the vicinity of the mixing grid of interest
169 (see again Figure 4 (c) and (d) for details of the mesh within the grid), but also comes with a number of
170 cells significantly higher than those proposed in the other simulation frameworks of the present paper (i. e.
171 around 200 million tetrahedral elements, for an equivalent global refinement level). The criterion expressed
172 in [7] of having 15 to 20 cells between two opposite walls is satisfied.

173 At this stage, no primastic layer can be implemented close to the walls with TrioCFD, yielding the need for
174 an active wall function almost everywhere in the model.

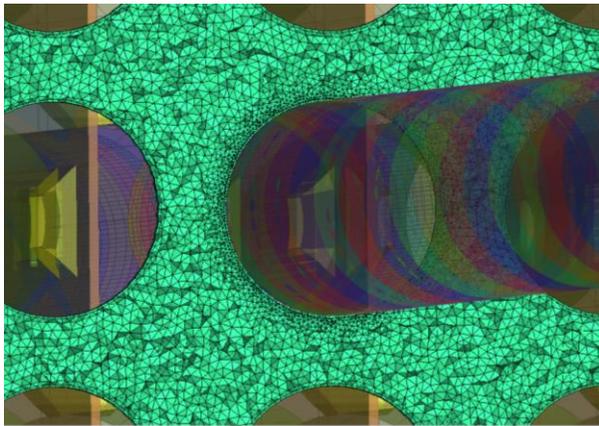
175



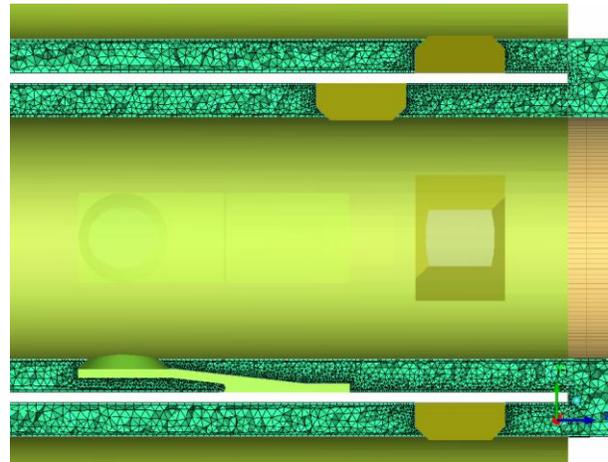
(a) EDF, *Code_Saturne*: mesh within the grid



(b) EDF, *Code_Saturne*: mesh in the bare bundle region



(c) CEA, *TrioCFD*: detail of the tetrahedral mesh within the bundle



(d) CEA, *TrioCFD*: detail of the mesh in the vicinity of springs and dimples within the grid

Figure 4. Views of the meshes for *Code_Saturne* and *TrioCFD*

176

177

178 3.3 Turbulence modeling and numerical settings

179 Framatome and EDF use the Smagorinsky subgrid-scale model [8] (in its standard version for Framatome
 180 and its dynamic one for EDF) to model unresolved scales, whereas CEA uses the WALE (Wall-Adapting
 181 Local Eddy-viscosity) model [9], designed to have a sub-grid scale viscosity with the right asymptotic
 182 behavior in the near-wall region.

183 The main features of the various numerical methods are given in Table 3.

184

Framatome	EDF	CEA
Spatial discretization		
Finite volumes with cells of any shape (trimmed/hexa for Framatome and hexa for EDF)		Specific Finite Element/Finite Volume for tetrahedral cells (P0/P1 for pressure, P1NC for velocity)
Numerical features for the unsteady solver		
Second order implicit temporal discretization.	<p>Smagorinsky constant varying from 0 to 0.065</p> <p>Velocity and pressure coupling ensured via a predictor-corrector algorithm with three outer sub-iterations every time step</p> <p>Centered time discretization (Crank-Nicolson and Adams-Bashford)</p> <p>Second order spatial discretization with implicit gradient reconstruction to take into account non orthogonal faces</p> <p>Centered convection scheme with 2% of upwind in order to smooth pressure and velocity oscillations in the streamwise direction.</p>	<p>2nd order Adams-Bashford explicit time integration</p> <p>Mixt 2nd order (centered and upwind) for convection, centered second order for diffusion</p>
Time-step and stability (CFL condition)		
Constant time-step of $5 \cdot 10^{-4}$ s Mean CFL about 0.95	Constant time step equal to 10^{-5} s. Mean CFL about 0.94 Maximum CFL about 2.25	Constant time-step equal to $6 \cdot 10^{-6}$ s Max CFL around 0.8
Parallel solution		
MPI	MPI & OpenMP [10]	Flat MPI

Table 3. Main numerical features in the different solvers

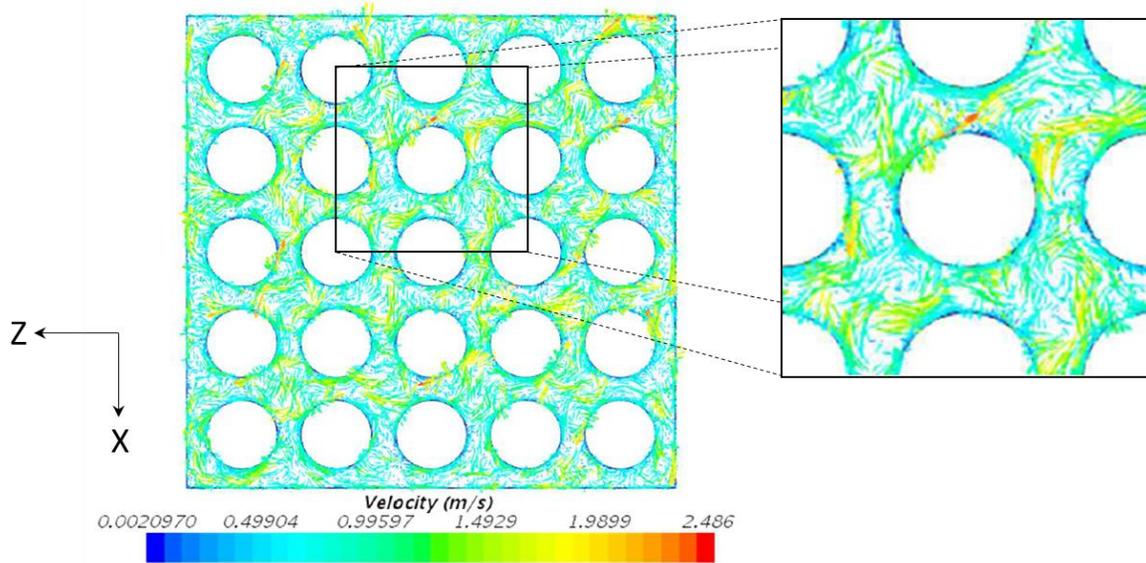
185

186

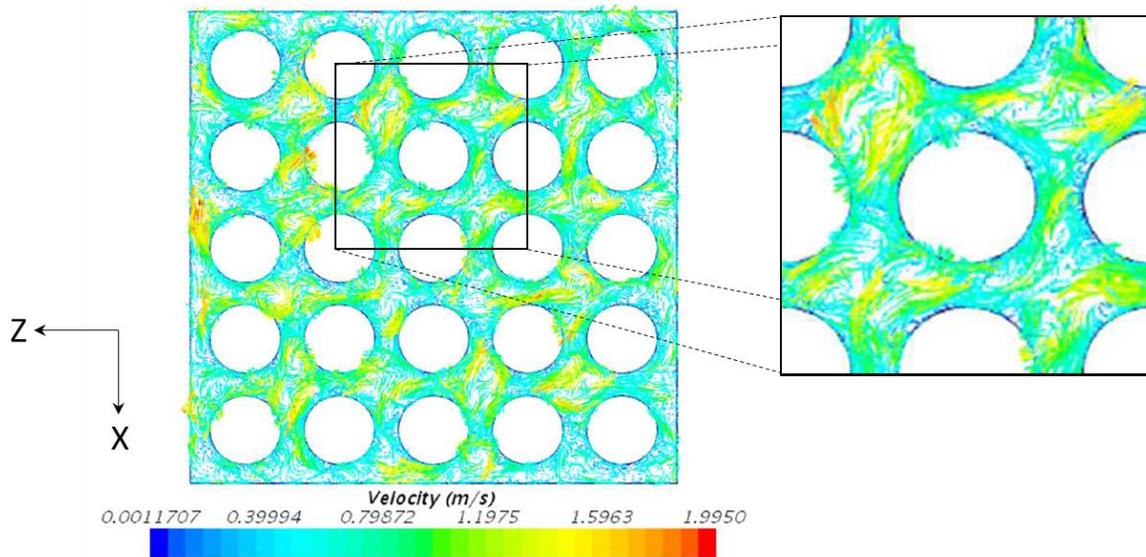
187

188 **4 RESULTS AND CROSS-COMPARISONS WITH RESPECT TO EXPERIMENT**

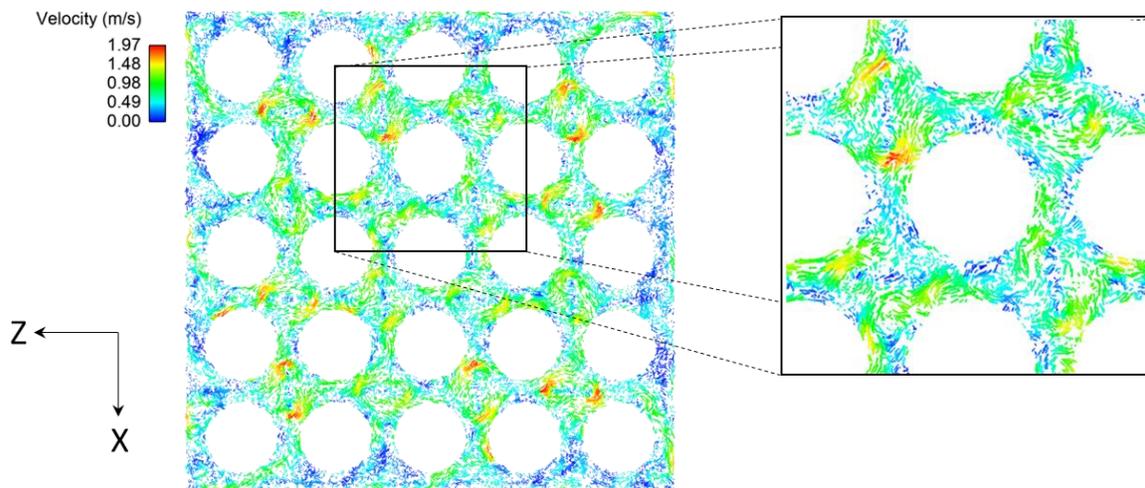
189 Figure 5 shows instantaneous 2D velocity vectors obtained with the different solvers 1 D_h and 2 D_h
190 downstream of the mixing vane grid. The secondary transverse flows show clearly that the mixing vane is
191 a split type one. The results are qualitatively in good agreement between the different models in terms of
192 both intensity and location of the generated flows.
193



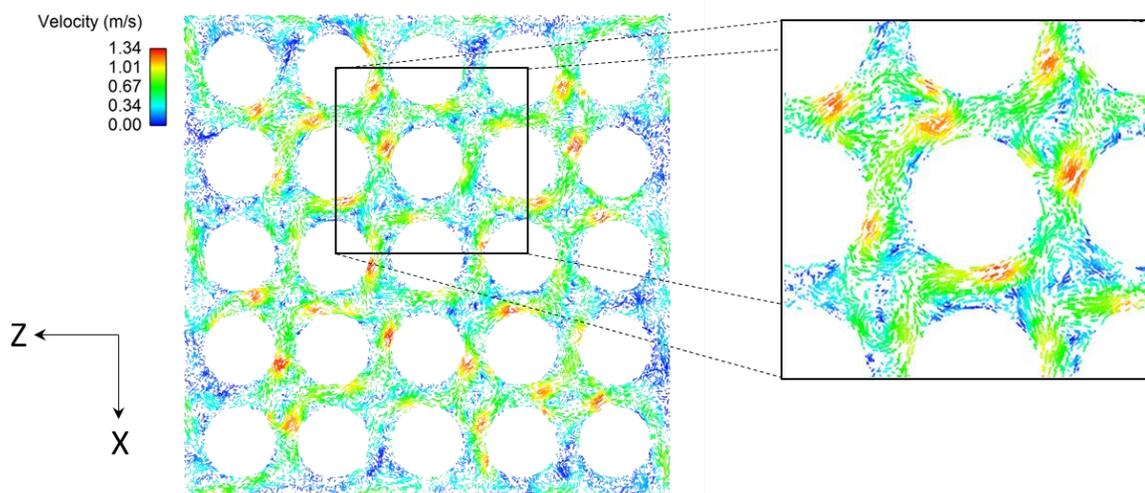
(a) Framatome, 1 D_h downstream of the grid



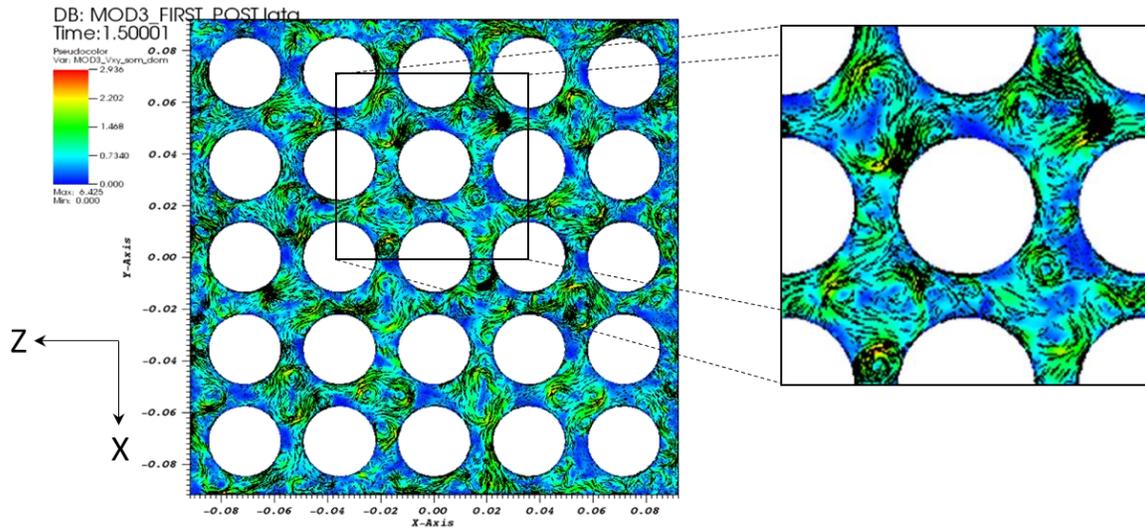
(b) Framatome, 2 D_h downstream of the grid



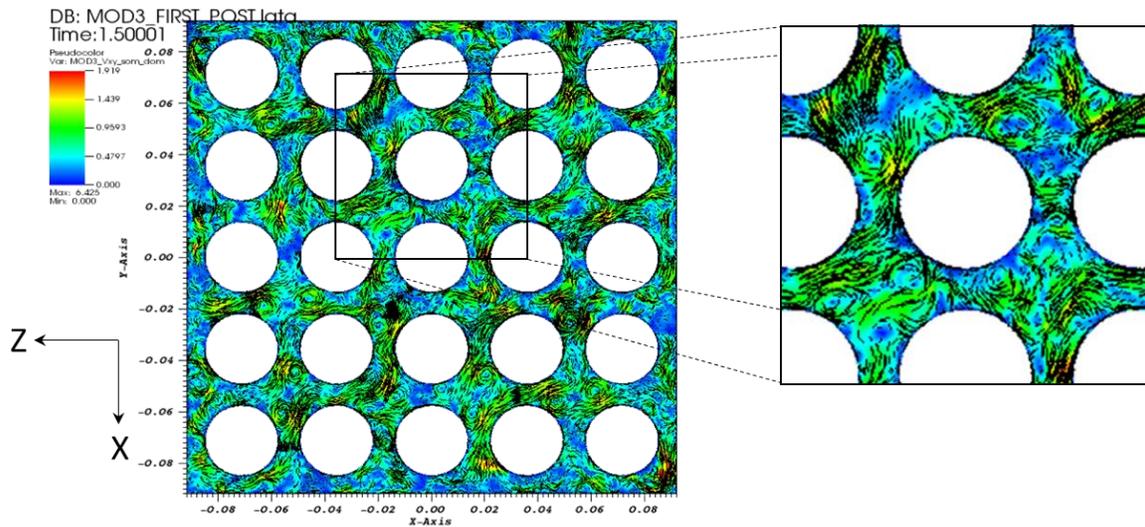
(c) EDF, $1 D_h$ downstream of the grid



(d) EDF, $2 D_h$ downstream of the grid



(e) CEA, 1 D_h downstream of the grid



(f) CEA, 2 D_h downstream of the grid

194 **Figure 5. Secondary transverse flow generated in the rod bundle downstream of the grid (colored**
 195 **by velocity in m/s)**

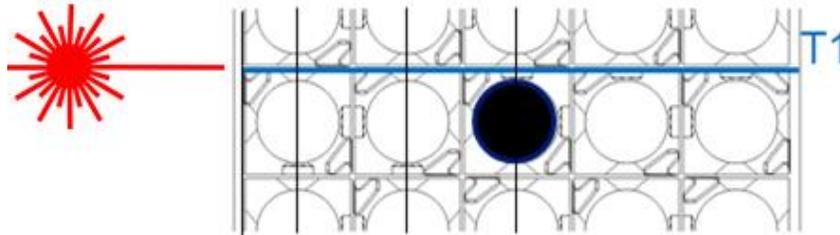
196

197 The next paragraphs focus on specifically measured quantities, to compare the three numerical solutions to
 198 experimental measurements.

199 **4.1 Velocity profiles downstream of the mixing grid**

200 Vertical and cross velocities predicted by Star-CCM+, *Code_Saturne* and TrioCFD are plotted along line
 201 T1 displayed on Figure 6. The numerical predictions compared to LDV data are shown in Figures 7 and 8,

202 at 2 and 5 D_h downstream of the mixing grid, respectively. The laser lights the window on the left. The
203 experimental profiles are only given for distances between 0 and 100 mm from sight window, due to
204 measurement difficulties further away from the window. The vertical lines correspond to the positions of
205 the rod symmetry axes.



207
208 **Figure 6. Location of line T1, laser is located on the left**

209
210 Whatever the CFD code used, the overall agreement between numerical and experimental data is very
211 satisfactory in the central region, for both components and both locations. Numerical results predicted by
212 TrioCFD and Star-CCM+ are closer to experimental vertical velocities, especially at 5 D_h for Star-CCM+,
213 whereas *Code_Saturne* is satisfactory at predicting the cross velocity. On the left side of the figures while
214 approaching the casing, differences of behavior can be observed between the three numerical predictions
215 and the experimental data due to the fact that the laser lights the window, which probably leads to distortion
216 of experimental measurements.

217 Concerning the cross velocity, it can clearly be observed a change of sign (between -0.5 and 0.5 m/s) at
218 D_h from the grid, which indicates a very marked influence of mixing vanes near the grid, clearly predicted
219 by the three CFD codes and measured by LDV. Further downstream at 5 D_h , the amplitudes of vertical and
220 cross velocity oscillations decrease: the influence of mixing vanes naturally decreases. The flow tends
221 toward a fully developed one in a bare bundle, after approximately 10 hydraulic diameters, although the
222 traces of the secondary vortices are still perceptible (not shown here).

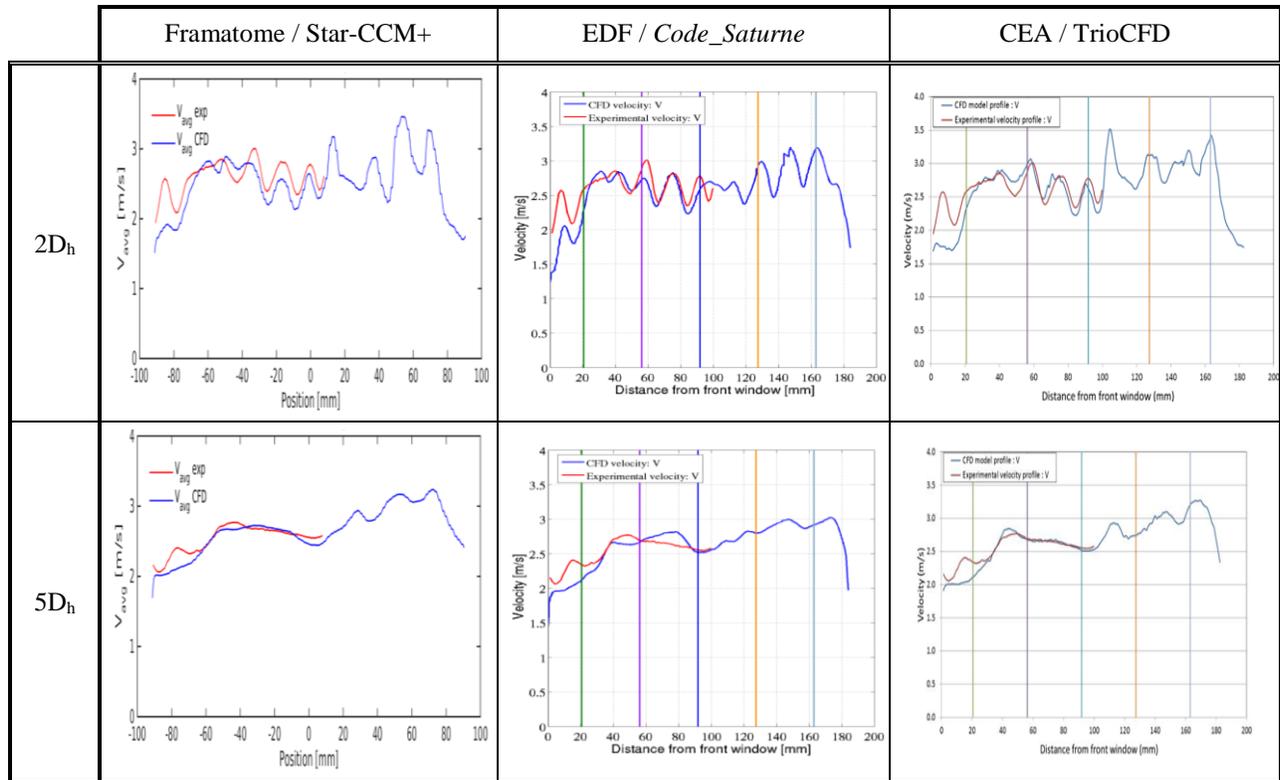
223

224

225

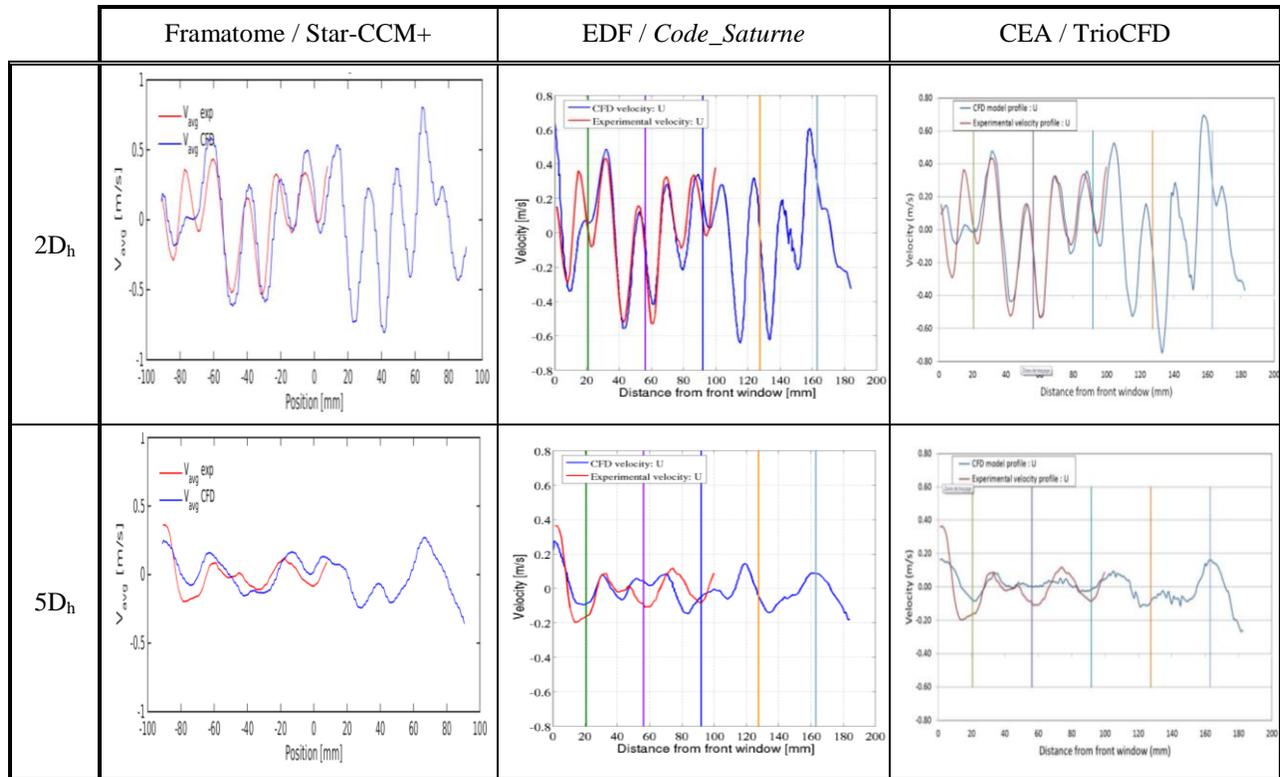
226

227



228 **Figure 7. Vertical velocity profiles at 2 and 5 D_h along T1 for the three CFD code (in blue CFD**
 229 **velocity and in red experimental velocity)**

230



231 **Figure 8. Cross velocity components profiles at 2 and 5 D_h along T1, for the three CFD codes (in**
 232 **blue CFD velocity and in red experimental velocity).**

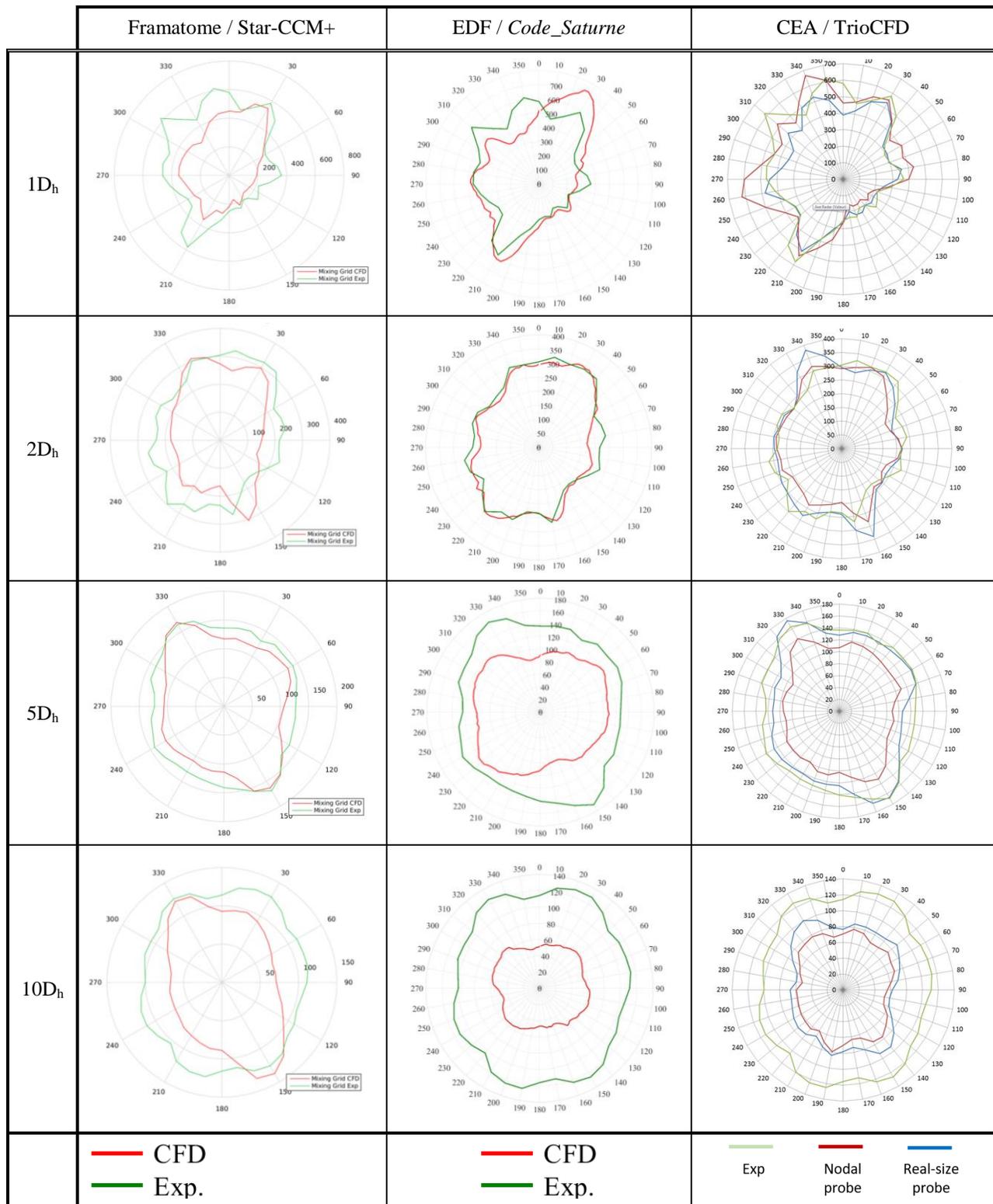
233

234 4.2 Pressure standard deviation distributions around the central rod

235 As stated in Section 2 and shown in Figure 2 for reference experimental results, unsteady pressure
 236 fluctuations measurements were performed around the central rod with a 10° increment. All three
 237 calculations were post-processed in order to compare the pressure standard deviation to experimental data
 238 thanks to polar representation. The comparison is presented at altitudes of 1 D_h, 2 D_h, 5 D_h and 10 D_h
 239 downstream of the mixing vane grid (see Figure 9).

240

241



242 **Figure 9. Polar representation of pressure standard deviation distribution around the central rod**
 243 **(red curves for CFD velocities, green curves for experimental velocities, blue curves do not have to**
 244 **be considered for this comparison)**

245 For CEA, numerical results are provided for both nodal pressure probes and so-called real-size probes
246 where the pressure is integrated over a surface representing the actual size of the experimental sensor. Only
247 the results corresponding to nodal pressure probes, will be considered for the following. Real-size probes
248 unexpectedly yield more filtered results, which has to be investigated in further work.

249 Results are here discussed in terms of azimuthal accuracy. Predictably, the pressure standard deviation
250 azimuthal pattern is heavily polarized immediately after the mixing grid. This polarization is well
251 reproduced by the codes in its (30° - 210°) diagonal, one hydraulic diameter downstream of the grid. The
252 standard deviation pattern grows into a more homogeneous shape with altitude as well as it reduces in
253 amplitude as the flow tends towards a fully developed flow in a bare bundle. All the solvers start drifting
254 away from the experimental results at 5 D_h , mostly in amplitude. Some (150° - 330°) polarization still
255 slightly persists at 5 D_h for Star-CCM+ and TrioCFD. This might be due to the use of wall functions for
256 TrioCFD or *Code_Saturne*. DNS or a wall-resolved LES in all programs would provide more elements to
257 conclude about this point.

258 **4.3 Pressure standard deviation decay with respect to the distance downstream of the** 259 **mixing grid**

260 The decay of the pressure standard deviation around the central rod downstream of the mixing grid is plotted
261 in Figure 10. The slope in the vicinity of the grid (up to 3 D_h downstream), where the standard deviation
262 are the highest, is rather correctly reproduced by all the models. This is not the case going further away
263 from the grid: if the change of slope around 4 D_h seen in the experiment is reproduced by the three
264 simulations, the numerical results are inaccurate in all cases, yielding a decay after 5 D_h significantly faster
265 than the experimental measurement.

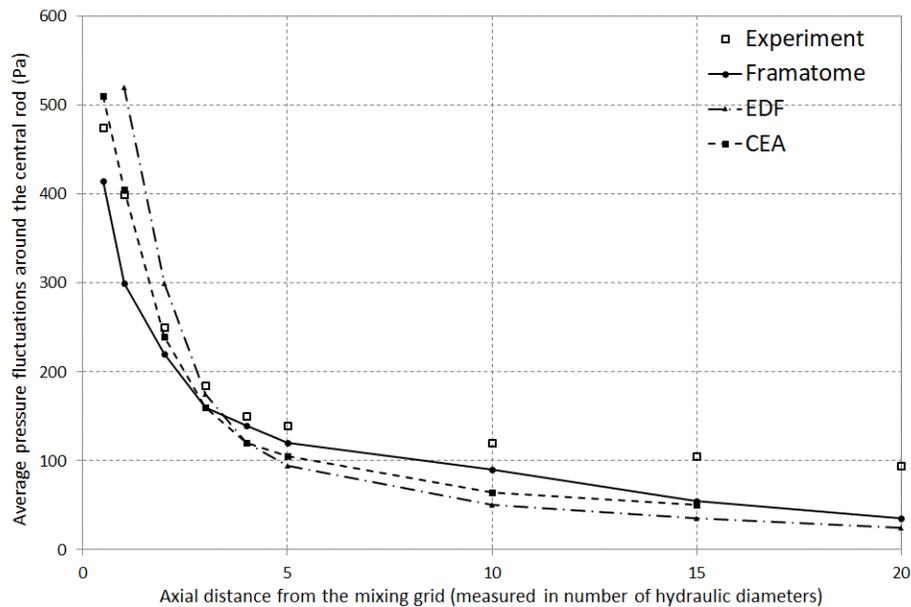
266

267 **5 DISCUSSION REGARDING LES VALIDATION AND RECOMMENDATIONS** 268 **FOR FUTURE WORK TO CONSOLIDATE THE PROPOSED RESULTS**

269 The simulations proposed in the current paper represent a significant computational effort to provide and
270 compare best-estimate solutions in the CALIFS 5x5 configuration. Each proposed computational model,
271 including its own meshing process, software choice and solver parameters, comes with some particular
272 hypotheses clearly described in sections 3.1, 3.2 and 3.3. Models of this size are built in agreement with
273 known recommendations, especially as far as mesh refinement is concerned and systematic *a posteriori*
274 sensitivity and convergences studies are classically out of reach for simple reasons of requested
275 computational power and time.

276 The first priority comment is that, whatever the mix of mesh characteristics and numerical solver, the
277 obtained results are closed to each other and to the experiments up to 4 Dh downstream of the mixing grid.
278 This applies to the azimuthal polarization of the pressure fluctuations where they are the most significant,
279 the accurate reproduction of secondary cross flows in the tube bundle and the decay of the pressure standard
280 deviation. The second comment is that all computational models fail at reproducing the correct decay of the
281 pressure fluctuations after 5 Dh downstream of the mixing grid.

282 While still not fully understood at this stage, this discrepancy in the agreement between experiment and
283 simulation seems to suggest that some persistent structures in the flow are missed in the numerical models,
284 which could be low frequency vortices generated by the vanes whose influence is masked by high
285 frequencies turbulent structures correctly captured just after the grid, or due to the use of wall functions
286 which filter the turbulence created by the walls. for *Code_Saturne* or TrioCFD
287



288

289

Figure 10. Pressure standard deviation decay downstream of the mixing grid

290

291 One strategy to address the remaining issue and go further the generally positive results obtained in the
292 proposed work would be to complement this study with sensitivity analyses to try to identify the modeling
293 components actually influencing the decay of the pressure fluctuations. The parameters to deal with could
294 principally be the mesh refinement, both in the wall normal direction and in the axial direction of the bundle,
295 and the models to account for the velocity profile in the boundary layer, up to a fully resolved and validated
296 solution close the walls to serve as a reference. This work should certainly be performed with less power

297 consuming models, obtained for instance after a reduction of the Reynolds number, to allow the efficient
298 production of series of calculation in a reasonable time. It obviously implies that some new experimental
299 results are obtained at these lower Reynolds numbers and that discrepancies of the same nature are still
300 observed in these less challenging conditions. Advanced hybrid RANS/LES approaches, as introduced in
301 paragraph 3.1, should be worth testing in this prospective work, especially in the case where the fully
302 resolved velocity field close to the walls proves preponderant.

303

304 **6 CONCLUSION**

305 The proposed advanced benchmark involving three CFD software solutions provides significant insights
306 related to the maturity of CFD simulation for single phase highly turbulent flows in rod bundles equipped
307 with one or more mixing grid(s), representative of the need for fretting assessment in PWR reactor and
308 other design applications involving in-core hydraulics.

309 One major lesson is that very satisfactory results are globally obtained with all the models independently
310 from the modelling options introduced above, suggesting that the software offer for this kind of calculations
311 has reached a generic and significant level of robustness, accuracy and maturity. This comes with a
312 mandatory prerequisite that the suitable expertise is mobilized in model building and tuning of numerical
313 parameters associated to the chosen method for time and space discretization.

314 Some discrepancies yet remain regarding the reproduction of the pressure standard deviation decay far
315 downstream of the mixing in the present case. It opens research topics to identify the origin of these
316 differences between simulation and experiment, especially to state if they are likely to reappear in other
317 configurations with more significant effects, like for the complex hydraulic situation at the bottom of a
318 PWR fuel assembly, with incoming jets from the main water supply impacting the nozzle with consequences
319 on the turbulent flow through the bundle and the first holding grid above.

320

321

322 **References**

323

- 324 1. "Review of Fuel Failures in Water Cooled Reactors", IAEA Nuclear Energy Series NF-T-2.1 (2010).
- 325 2. M.P. Païdoussis, "A review of flow induced vibrations in reactors and reactor components. Nuclear Engineering
326 and design 74, 31-60 (1982).
- 327 3. F. Moreno, B. Collard, V. Faucher, "Measurement of fluctuating fluid pressure exerted on the walls of tube bundle".
328 Flow Induced Vibrations, The Hague, Netherlands, 2016.

- 329 4. F. Moreno, S. Bantiche, F. Bazin, T. Lohez, D. Picard, S. Testanière, L. Rossi, “Unsteady Pressure and Velocity
330 Measurements in 5x5 Rods Bundle Using Grids With and Without Mixing Vanes”. 12th International Topical Meeting
331 on Nuclear Reactor Thermal-Hydraulics, Operation and Safety, Qindao, China, 2018.
- 332 5. P. Sagaut, “Large Eddy Simulation for Incompressible Flows (Third ed.)”. Springer, 2006.
- 333 6. S. Delafontaine, “Simulation of unsteady fluid forces on a single rod downstream of mixing grid cell”. Nuclear
334 Engineering and Design, vol. 332, 2018.
- 335 7. Best practice Guidelines for the use of CFD in Nuclear Reactor Safety Application, Collective document, 2007,
336 OECD Nuclear Energy Agency, Committee on the Safety of Nuclear Installations, NEA/CSNI/R-2007-5.
- 337 8. M. Germano, U. Piomelli, P. Moin, W. Cabot, “A dynamic subgrid-scale eddy viscosity model”, Physics of Fluids,
338 vol. 3(7), 1991.
- 339 9. F. Ducros, F. Nicoud, “Subgrid-scale stress modelling based on the square of the velocity gradient tensor”. Flow
340 Turbulence and Combustion, vol. 62(3), 1999.
- 341 10. Y. Fournier, J. Bonelle, C. Moulinec, Z. Shang, A.G. Sunderland, J.C. Uribe, “Optimizing Code_Saturne
342 computations on Petascale systems”. Computers & Fluids, vol. 3(7), 2011.
- 343 11. Benhamadouche, S., Moussou, P., Le-Maître, C., 2009. CFD estimation of the flow-induced vibrations of a fuel
344 rod downstream a mixing grid. Proceedings of PVP 2009 ASME Pressure Vessels and Piping 2009 / Creep 8
345 Conference, July 22-26, Prague, Czech Republic.
- 346 12. D. Wells and Y. Hassan, “Overview of CFD Round Robin Benchmark of the High Fidelity Fuel Rod Bundle
347 NESTOR Experimental Data”, The 16th International Topical Meeting on Nuclear Reactor Thermal Hydraulics
348 (NURETH-16) , Hyatt Regency Chicago, Chicago, IL, USA, August 30-September 4 (2015).
- 349 13. S. Benhamadouche. On the use of (U)RANS and LES approaches for turbulent incompressible single phase flows
350 in nuclear engineering applications. *Nuclear Engineering and Design*, 312, pp. 2-11 (2017)
- 351 14. Piomelli, U., 2008. Wall-Layer Models for Large-Eddy Simulations. Progress in Aerospace Sciences, Large Eddy
352 Simulation - Current Capabilities and Areas of Needed Research, 44, no. 6 : 437–46.
- 353 15. Benhamadouche, S., Gauffre, M.-C., Badel, P. Wall-modelled large eddy simulation of the flow through PWR fuel
354 assemblies AT ReH=66,000 - Validation on califs experimental setup, International Topical Meeting on Advances in
355 Thermal Hydraulics, ATH 2018 - Embedded Topical Meeting pp. 765-776, 2018.
- 356 16. U. Bieder, F. Falk, G. Fauchet, Les analysis of the flow in a simplified pwr assembly with mixing grid, *Progress*
357 *in Nuclear Energy* 75 (2014) 15 – 24.
- 358 17. G. Chen, Z. Zhang, Z. Tian, X. Dong, Y. Wang, Cfd simulation for the optimal design and utilization of experiment
359 to research the flow process in pwr, *Annals of Nuclear Energy* 94 (2016) 1 – 9.
- 360 18. J. Xiong, R. Cheng, C. Lu, X. Chai, X. Liu, X. Cheng, Cfd simulation of swirling flow induced by twist vanes in
361 a rod bundle, *Nuclear Engineering and Design* 338 (2018) 52 – 62.
- 362 19. U. Bieder, F. Falk, G. Fauchet, Cfd analysis of the flow in the near wake of a generic pwr mixing grid, *Annals of*
363 *Nuclear Energy* 82 (2015) 169 – 178.
- 364 20. Z. Karoutas, C. Gu, B. Scholin, 3-d flow analyses for design of nuclear fuel spacer, Proceedings of the Seventh
365 International Meeting on Nuclear Reactor Thermal-Hydraulics (1995) 3153–3174.

- 366 21. W.-K. In, D.-S. Oh, T.-H. Chun, Flow analysis for optimum design of mixing vane in a pwr fuel assembly, Nuclear
367 Engineering and Technology 33 (2001).
- 368 22. S. Cheng, H. Chen, X. Zhang, Cfd analysis of flow field in a 5x5 rod bundle with multi-grid, Annals of Nuclear
369 Energy 99 (2017) 464 – 470.
- 370 23. J. R. Lee, J. Kim, C.-H. Song, Synthesis of the turbulent mixing in a rod bundle with vaned spacer grids based on
371 the oecd-kaeri cfd benchmark exercise, Nuclear Engineering and Design 279 (2014) 3 – 18.