Uniform flow around a square cylinder using the Self-induced angular Moment Method turbulence model

*Johansson, Jens\textsuperscript{1)}, Nielsen, M. P.\textsuperscript{2)}, Nielsen, Leif O.\textsuperscript{3})

\textsuperscript{1) Institute of Technology and Innovation, University of Southern Denmark, Campusvej 55}
\textsuperscript{2) Alectia A/S, Teknikerbyen 34, 2830 Virum, Denmark}
\textsuperscript{3) Department of Civil Engineering, Technical University of Denmark, Anker EngelundsVej 1, 2800 Kgs. Lyngby, Denmark}
\textsuperscript{1) jensj@iti.sdu.dk}

ABSTRACT

The uniform flow around a square cylinder at Reynolds number 1e5 is simulated in a three-dimensional domain by means of the newly developed Self-induced angular Moment Method, SMoM. The model does not utilize Reynolds averaging. No additional transport equations are introduced and no implicit or explicit filtering is performed. The model is, in all its simplicity, a modification of the classical constitutive equations of fluids to which a term is added that accounts for the transfer for angular momentum between parts of the fluid. The time-mean and fluctuating force coefficients, pressure distributions and velocity fields have been determined along with circumferential surface pressure correlations. All simulated quantities have been compared to experimental findings and state-of-the-art Large Eddy Simulations, LES. No LES simulations could be found in literature, which provided results at the same level of accuracy for all the reported parameters. Hence, the SMoM turbulence model is found to provide an alternative to the Large Eddy Simulations.

1. INTRODUCTION

The flow around a square cylinder obviously involves very simple geometry in a civil engineering context. However, the flow around the square cylinder is highly complex. For this reason the flow over a square cylinder has previously been used as a test case for Large Eddy Simulations. The flow was chosen as a test case at a workshop in Rottach-Egern, Germany in the summer of 1995. The findings from the workshop were published in (Rodi et. al 1997). Ten different groups submitted a total of sixteen sets of results. As there were a large variation of numerical methods it was difficult to evaluate the actual performance of the individual method. Unfortunately, this lead to the somewhat weak conclusion that “no conclusion could be made on the effect of choice of numerical method”. This was later addressed by (Nakayama and Vengadesan 2002). No single set of results was highlighted, and the results of the workshop
were concluded with; “Overall, it is our judgment that no single simulation is uniformly good; significant faults can be found in every submission”.

In September 1996 the flow around a square cylinder was also adopted for the Second ERCOFTAC Workshop on Direct and Large-Eddy Simulation held in Grenoble, France. The proceedings of this workshop were published in (Voke 1997). Here seven groups of researchers submitted a total of 20 data sets, of which a selection was compared in (Voke1997). Grid convergence was considered in some of the submitted data sets, though it was stated that none had demonstrated convergence convincingly. It was noted that the simultaneous prediction of the recirculation length and the recovery of the centerline velocity was very difficult, as was also noted in the (Rodi et al. 1997) workshop. One group defied this rule, and predicted both. However, their reported drag coefficient was not reassuring. The NT7 submission was highlighted as this simulation predicted most of the integral parameters correctly, though there were some discrepancies in the velocity field. The force coefficients predicted by this group are provided in Table 2. The final conclusion at the (Rodi et al. 1997) workshop was consistent with previous findings; “The flow over and following the square cylinder is therefore proving a major challenge to current LES techniques, a stimulus to further investigation and a problem to which, at present, we cannot claim to have a definitive solution.”.

(Lee and Bienkiewicz1998) performed 2D and 3D LES simulations at $Re = 22000$ using isoparametric elements in a Galerkin FEM discretization. The standard Smagorinsky model was employed using $C_s = 0.1$. The global force statistics were well predicted, as seen in Table 2. However the velocity profiles downstream of the cylinder were over predicted, also causing a slight over prediction of the recirculation length. The time-mean pressure distribution agreed well with the pressure distribution reported by (Murakami and Mochida 1995) who used a dynamic Smagorinsky model, and also agreed well to the experimental findings by (Bearman and Obasaju1982).

(Sohankar et al. 2000) later performed six different simulations using three different sub-grid scales models. One simulation was performed with each of the Standard Smagorinsky ($C_s = 0.1$) and the dynamic Germano-Lilly model. The remaining four simulations were performed with a dynamic one-equation model developed by Davidson. See (Sohankar et al. 2000). Blockage corrected results of one of the simulations were presented, which agreed very well with the corrected global quantities from experiments. The effect of span-wise length was investigated. The fluctuating drag were reduced by 6% by extending the cylinder length from four to seven side lengths, while other quantities were within ±2%. For an increase in time resolution by a factor 2 the only significant change was a 5% reduction of the fluctuating drag. Further the rms quantities $(\overline{u'v'})^{1/2}$, $(\overline{v'w'})^{1/2}$ in the centerline were very well predicted. However, the peak in $(\overline{v'w'})^{1/2}$ occurred early as was also mentioned in (Voke 1997), and the cross sectional profiles of velocity variances (rms squared) showed some deviation from the experimental findings for one of their downstream locations. The reported recirculation length was smaller than the experimental, and the velocity in the recirculation zone was not found to be as negative.

In the present paper the case of flow around a square cylinder illustrated in Fig. 1 will be used to assess the performance of the SMoM turbulence model. We consider a square cylinder of side length $d$ and a span-wise length $l = 2d$ in the $z$-direction. The cylinder is subjected to turbulence-free incoming stream of a velocity $u_\infty$ in the $x$-direction. The dimensions of the computational domain are given the cross-stream height $H = 20d$ in the $y$-direction and stream-
wise length $L = 40d$, with the inlet plane placed an upstream distance of $10d$ from the cylinders center of gravity.

Fig. 1. – Principal orientation of the square cylinder in the computational domain.

2. THE SMoM TURBULENCE MODEL

The SMoM turbulence model has been suggested by (Nielsen et al 2006). The model was further calibrated and boundary conditions were developed in (Johansson, J. 2011). A short introduction to the model is offered here.

If we let the axis $x_1, x_2,$ and $x_3$ define a right-hand coordinate system, then the Navier-Stokes equation in continuity form, with zero body forces, may be written in index notation as in Eq. (1).

$$
\rho \frac{\partial u_i}{\partial t} + \rho \frac{\partial}{\partial x_j} (u_j u_i) = \frac{\partial \tau_{ij}}{\partial x_j}
$$

where $t$ is time, $\rho$ is density, $u_i$ is the velocity vector, and $\tau_{ij}$ is the Cauchy stress tensor. The SMoM turbulence model is a zero equation turbulence model in the sense that no additional transport equations are introduced. The model is merely a modification of the constitutive equation of the Newtonian fluid. For the Newtonian fluid the classical terms takes into account the transfer of linear momentum between parts of the fluid. In the SMoM turbulence model a term is added which takes into account the transfer for angular momentum between parts of the fluid. Thus the constitutive equation of the fluid reads

$$
\tau_{ij} = -p \delta_{ij} + 2\mu \delta_{ij} + 2b u^a \Omega_{ji}
$$
where $d_{ij}$ is the strain rate tensor, $\delta_{ij}$ is Kronecker delta, $\mu$ is the dynamic viscosity, and $p$ is the pressure. The first two terms completes the usual constitutive equation for the Newtonian fluid. The term $2b_u^a \Omega_{ij}$ is thus the added term, where $\Omega_{ij} = 1/2\left( \partial u_i / \partial x_j - \partial u_j / \partial x_i \right)$ is the rate-of-rotation tensor, $b$ is the local value of SMoM models scale parameter and $u^a$ is the absolute velocity $\sqrt{u_i u_i}$ to the power of $\alpha$, which is a model parameter. The local value of $b$ is given by boundary conditions and the base value of the scale parameter $b_0$, within the case of smooth walls may be determined from

$$b = k_1 \left( \frac{\mu}{\rho} \right)^{1-\alpha} r_p^\alpha \rho$$

(3)

The value of the model parameters $k_1 = 0.0158$ and $\alpha = 0.8482$ have been determined from a least square fit of analytical velocity profiles in smooth pipes to the experimental data in (Nikuradse 1932). The constant $r_p^\alpha$ is labeled the scale factor, which is not a fluid property, but rather a flow property. The scale factor represents a characteristic length-scale in the simulated flow. In the case external flow around a square cylinder we set the length $r_p$ equivalent to the cylinder side length $d$.

3. SIMULATIONS

The simulations have been performed using the open source software package OpenFOAM version 1.6.x, which is based on the Finite Volume Method. An existing implementation of the PISO solution method was used through a modification of an existing solver. The Crank-Nicholson scheme was used for temporal discretization. For the spatial discretization second order central differencing was used, while a built-in Total Variation Diminishing, TDV, scheme was used for the convective terms. At the outlet boundary a built-in advective boundary condition was used, which is based on $d u_i / d t + u_i \cdot d u_i / d x = 0$. For the upper, lower, and cylinder-end boundaries a symmetry boundary condition was used. At the cylinder surface the no-slip condition was enforced, and the scale parameter $b$ was given a linear variation with the distance to the wall within a zone around the square extending to an absolute distance of $y_T = 0.2d$ away from the surface. Hence at the surface of the cylinder $b = 0$ while $b = b_0$ at, and beyond the transition point $y_T$. In the implementation of the SMoM model the scale parameter $b$ was considered to be a local constant in the sense that the scale parameter $b$ was moved outside the differentiation indicated in Eq. (1). With this rather crude handling of the boundary conditions an accurate determination of the surface shear stresses cannot be expected. Hence, further work on the boundary conditions is still in progress. The computational grid was structured according to the general layout outlined in Fig. 2, with a grid refinement zone with a diameter of $5d$. Three different computational grids were used; Grid A, B and C. The grids were designed as structured grids based on the number of cells listed in Table 1.
4. RESULTS AND DISCUSSION

Simulations were allowed to reach a “stable” condition before any results were collected. The results presented here are based on a non-dimensional time interval of minimum $\Delta t u_\infty / d = 400$. Both spatial and temporal convergence has been considered. The time-mean force coefficients and Strouhal numbers are provided in Table 2 for the three levels of grid refinement and time-step resolutions. In the following, the velocities are labeled $u$, $v$, and $w$ for $u_1$, $u_2$, and $u_3$, respectively. Further non-dimensional time-steps $\delta t^* = u\delta t / d$, and velocities e.g. $u/u_\infty$ are used.

One fundamental problem of a grid convergence study is the rapid growth of computational demands that follow from grid refinement. Good practice might be that at least three different grid resolutions, with each having twice the amount of cells in all directions as its predecessor. In this sense, a grid convergence study has not been performed here. However, three different grids with successively finer cells have been used. As shown in Table 1 the number of cells in the span-wise ($z$) direction was kept unchanged at the second step of refinement. This was also the case for the number of cells in the outer quadrilateral parts of the domain. However, the number of cells was doubled in the grid refinement zone’s $x,y$-plane.

4.1 Global force statistics

Fig. 3 shows a time-history plot of the global lift and drag force coefficients $C_l$ and $C_d$, respectively. The vertical lines indicate the starting point of the time-averaging period for the simulation shown. The time averaging period was chosen from a purely visual inspection of the drag time history. Within the time averaging period the simulation is seen to be at a reasonably...
stable level. The time-averaged coefficients for lift, $C_l$, drag, $C_d$, and the root mean square of their fluctuations, i.e., $C'_l$ and $C'_d$, as well as the Strouhal numbers $St$, are listed in Table 2 together with results from published findings. The drag coefficients are a sum of the contributions from pressure drag and friction drag. In the simulation based on grid C the individual contributions were calculated, and the friction drag was found to contribute with less than 1% of the total drag. The Strouhal number is based on the peak in the spectrum of the lift force shown in Fig. 4.

![Fig. 3. Non-dimensional time series of the drag and lift coefficients for the present simulations. Here shown for Grid A.](image)

To evaluate grid convergence the global force statistics were compared for the three grid resolutions. The results are shown in Table 2. A 5% decrease in time-mean drag can be observed when refining the coarse grid (grid A) twice to obtain the finest grid (grid C). At the same time a 33% increase in the fluctuating drag $C'_d$ and an 22% decrease in the fluctuating lift can be observed accompanied by a 3% decrease in Strouhal number. It appears that the Strouhal number is easily predicted, as the three levels of grids give almost identical results, which are also within the experimental range. This observation is consistent with the notion of (Rodet al. 1997), that the Strouhal number was only weakly affected by the combination of numerical methods. The finer grids give a time-mean drag coefficient $\bar{C}_d$, that is less than 2% higher than the reported experimental range, and a fluctuating drag coefficient $C'_d$ that is within the reported experimental range.

As the rate of change in fluctuating drag from grid B to C (+20%) is actually higher than the change when going from grid A to grid B (+10%), the data does not suggest that the solution has converged. However, both fluctuating lift and time-mean drag have reduced rates of change at the second step of refinement. The fluctuating lift goes from a 13% increase to a 7% increase, while the time-mean drag goes from a 3.4% decrease to a 1.8% decrease. Hence, the latter findings do suggest that the results are converging. The large change in the fluctuating quantities may be related to the different levels of time steps used. One could speculate that the smaller time steps caused an increase in the fluctuations, as is the case with the present data. This issue is addressed in the following.

To investigate temporal convergence, simulations have been carried out with three different time-steps. All three simulations are based on grid B. Again we use the global force statistics to
evaluate the convergence of the results. Table 2 lists the global force statistics and Strouhal numbers. The Courant-Friedrich-Levi, CFL, condition is enforced in all the simulations i.e. Courant numbers are kept below 1.0. The non-dimensional time steps used were $\delta t^* = 0.008; 0.004$ and $0.002$ which resulted in Courant numbers of approximately $Co = 0.5, 0.25$ and $0.125$ respectively. An increase in the temporal resolution is seen to cause an increase in the time-mean drag, however the increase in temporal resolution from $\delta t^* = 0.008$ to $\delta t^* = 0.002$ only caused a 3% change in the time-mean drag, while other parameters remained virtually unchanged. Hence, in the following we shall assume that temporal convergence has been reached. We will also conclude that the strong changes in the fluctuating drag experienced when refining the grid are caused by the grid, and hence not by the altered time-step.

<table>
<thead>
<tr>
<th>Source</th>
<th>$C_d$</th>
<th>$C'_d$</th>
<th>$C'_l$</th>
<th>$St$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid A, $\delta t^* = 0.008$</td>
<td>2.36</td>
<td>0.127</td>
<td>1.12</td>
<td>0.134</td>
</tr>
<tr>
<td>Grid B, $\delta t^* = 0.008$</td>
<td>2.24</td>
<td>0.14</td>
<td>1.27</td>
<td>0.134</td>
</tr>
<tr>
<td>Grid B, $\delta t^* = 0.004$</td>
<td>2.28</td>
<td>0.14</td>
<td>1.27</td>
<td>0.134</td>
</tr>
<tr>
<td>Grid B, $\delta t^* = 0.002$</td>
<td>2.31</td>
<td>0.15</td>
<td>1.26</td>
<td>0.134</td>
</tr>
<tr>
<td>Grid C, $\delta t^* = 0.002$</td>
<td>2.24</td>
<td>0.17</td>
<td>1.37</td>
<td>0.130</td>
</tr>
<tr>
<td>Grid C, $\delta t^* = 0.002$</td>
<td>2.07(b)</td>
<td>0.16(b)</td>
<td>1.27(b)</td>
<td>0.125(b)</td>
</tr>
<tr>
<td>Num, (Murakami and Mochida 1995)</td>
<td>2.09</td>
<td>0.12$^{(a)}$</td>
<td>1.60$^{(a)}$</td>
<td>0.132</td>
</tr>
<tr>
<td>Num, (Yu and Kareem 1997)</td>
<td>2.14</td>
<td>0.25</td>
<td>1.15</td>
<td>0.135</td>
</tr>
<tr>
<td>Num, (Voke 1997), Group NT7</td>
<td>2.05</td>
<td>0.12</td>
<td>1.39</td>
<td>0.131</td>
</tr>
<tr>
<td>Num, (Lee and Bienkiewicz 1998)</td>
<td>2.06</td>
<td>0.214</td>
<td>1.214</td>
<td>0.134</td>
</tr>
<tr>
<td>Experiments</td>
<td>2.05-2.2</td>
<td>0.17-0.22</td>
<td>1.2-1.32</td>
<td>0.118-0.142</td>
</tr>
</tbody>
</table>

**Table 2.** Global force coefficients and Strouhal numbers for three different grid resolutions and different time-steps. The listed experimental range is based on the published experimental data in (Vickery 1966), (Lee 1975), (Bearman and Obasaju 1982), (Norberg 1993), and (Lyn et al. 1995). Notes: $^{(a)}$ Phase averaged results; $^{(b)}$ blockage corrected using a simplification of (Maskell 1963) shown in (Sohankar et al. 2000).

Grid C was the finest grid used in the current simulations. This grid is seen to reproduce the experimental values very well. The largest deviation from the experimental values is approximately 5%. Other values are within the reported experimental range. This level of accuracy is sufficient for civil engineering purposes. The effect of applying blockage corrections is shown in Table 2. The simplified application of (Maskell 1963) in (Sohankar et al. 2000) has been used. The blockage correction is given by $\alpha_q = (1 - 1.5\beta)^{-1}$ where $\beta$ is the blockage ratio, $C_d/C_{d,c} = \alpha_q$ and $C'_d/C'_{d,c} = C'_l/C'_{l,c} = \alpha_q$. The Strouhal number was corrected using $St_c = St/\sqrt{\alpha_q}$. Except for the Grid C results in Table 2 no blockage correction has been applied to the remaining results presented in the following. This decision was based on the fact, that the method is yet to be validated for the present numerical setup. The spectral content of the global forces has also been investigated and compared to the results reported by (Vickery 1966). Fig. 4 shows the reduced normalized power spectrum of the lift and drag forces in the case of simulation on grid B using $\delta t^* = 0.004$. The lift spectrum is seen to
resemble the experimental curve obtained by (Vickery 1966). Both the variation (shape) and peak value is in very close agreement; however the peak is shifted slightly towards higher frequencies i.e. a higher Strouhal number is obtained in our simulations. The Strouhal number obtained is still within the experimental range, as indicated in Table 2. The shape of the spectrum of the drag force is also seen to be very well resembled, though the energy seems to be narrower banded in the present simulations. When comparing these curves one should keep in mind that the inlet flow in the simulations was completely turbulence free, whereas some inlet turbulence has certainly been present in the experiments by (Vickery 1966). Such inlet turbulence is likely to be high frequency, and hence one could imagine this to widen the drag spectrum (distributing energy over a wider range of frequencies), and consequently reducing the spectrum peak value. The spectra's presented was calculated based on Welch's averaged periodogram method (Welch 1967) using a hamming window, with a window size of one quarter of the time signal and the overlap was half the window size.

![Graph](image)

**Fig. 4.** The reduced normalized spectrum of lift- (left) and drag force (right) coefficients. , (Vickery 1966); , Present simulations, grid B.

### 4.1 The velocity field

Velocity profiles may contribute to the understanding of the flow around bluff bodies. In some fields of civil engineering the velocity profiles might even be the primary goal. These can for instance be used for comfort studies around buildings, or in the evaluation of dispersion of pollutants etc.

In the present work we will include the velocity profiles to ensure that the simulated flow fields match that of corresponding experiments. The velocity profiles obtained in the present simulations are compared to the results of (Lyn et al. 1995). The data used in the plot are from the ERCOFTAC database. Details of the velocity field are presented at a number of cross flow stations positioned at; $x/d = 0, x/d = 0.5, x/d = 1.0$ and $x/d = 4.5$. Furthermore, details are given in a stream-wise line placed in the center of the cylinder $y/d = z/d = 0$. 
In the following figures the velocity profiles are shown for the three different simulations in the spatial convergence study. I.e. the velocity profiles are shown for grid A, B, and C\textsuperscript{1}. The recirculation length\textsuperscript{2} is found to be 1.36, 1.34 for grid A and B, respectively, while (Lyn et al. 1995) reported 1.38. The maximum time-mean reverse velocity in the recirculation zone was found to be $-0.183$, $-0.153$ for grid A and B, respectively, while (Lyn et al. 1995) reported it to be $-0.211$. The velocity at $(x/d = 10)$ has recovered to $0.78 (0.68$ at $x/d = 30$) and 0.64 ($0.71$ at $x/d = 30$) for grid A and B, respectively. The recirculation length is well captured with both grid A and B. (Under-predicted by less than 3\%)

![Fig. 5. Time-mean stream-wise velocity. (Lyn et al. 1995). Present simulations; —, Grid A; --, Grid B.](image)

At first glance of Fig. 5 the prediction of the recovery velocity seems to depend on the resolution of the outer grid, i.e. the successive finer grids provide improved predictions. However, the final recovery velocity at $x/d = 30$ is very similar for the two grids. The simulation based on grid B seems to show a peak in the recovery velocity centered around a downstream location of $x/d \approx 4.0$. However, this “peak” value is slightly lower than the final recovery. The time-mean stream-wise and cross-stream velocities are shown in Fig. 6 for each of the four cross-stream locations. Though no quantitative measures of divinations have been taken, it seems that refinement of the grid leads to predictions in better agreement with the experiments. We note that the simulations provide time-mean velocity profiles which are hardly distinguishable from the experimental findings.

The fluctuating quantities are given in terms of the Reynolds stress components\textsuperscript{3}. Fig. 7 shows the quantities at the four cross-stream locations. It is obvious that the finer grids, again, provide profiles in better agreement with the experimental findings. Finally, Fig. 8 shows the Reynolds stress components measured in a stream-wise line at $y/d = z/d = 0$. We note again that the finer the grid, the better agreement with experimental findings. In (Voke 1997) it was noted that the downstream location of the peak of $\overline{v'v'}$ found in the

\textsuperscript{1}The grid C data is unfortunately incomplete due to data corruption during post-processing.
\textsuperscript{2}When measured from $x = 0$.
\textsuperscript{3}Or rather the specific Reynolds stress components, as they have not been multiplied by the density, $\rho$. The measurements are basically velocity correlations, but are often referred to as the Reynolds stress. This is due to their general appearance in the Reynolds Averaged Navier-Stokes equations.
numerical simulations, was closer to the cylinder than found in the experiments. The same
tendency can be observed in the data by (Sohankaret al. 2000).

Fig. 6. Time-mean velocity distributions at four cross flow stations, (Lyn et al. 1995). Present 
simulations: —, Grid A; ——, Grid B; ———, Grid C. (a) is the stream-wise velocity \( \bar{u} \); (b) is the 
cross-stream velocity \( \bar{v} \).

The present data seems to support the experimental findings. Overall, the velocity distributions,
including their fluctuating quantities, have shown close agreement with experimental findings. 
Refinement of the grid has led to better agreement with experimental findings, especially for 
the fluctuating quantities. In regard of convergence of the results, it seems that (without quantitative 
measures to support this) the results are converging.
Fig. 7. The Reynolds stress terms at four cross flow stations, \( \times \) (Lyn et al. 1995). Present simulations: ---, Grid A; ---, Grid B; ---, Grid C. (a) is the \( u' \) component; (b) is the \( v' \) component; (c) is the cross correlation between \( u' \) and \( v' \).
4.2 Surface pressure statistics

Here we will present the pressure distributions as time-mean local pressure coefficients $C_p$. The pressure distributions will be compared to the experiments by (Lee 1975) and (Bearman and Obasaju 1982). The fluctuating pressure coefficients $C'_p$ are also presented. The time-mean pressure distributions obtained in the numerical simulations have been corrected, so that $C_p = 1.0$ at the front center surface. The time-mean pressure distributions are given in Fig. 9. The sharp corner points of the square pose a problem in the numerical simulations. The singularities here cause the pressure readings to deviate substantially. As the grid is refined this effect is minimized. Using the coarse grid leads to a pressure distribution that deviates slightly (say 5-10%) on the top and bottom surfaces of the cylinder. However, as the grid was refined the pressure distributions are found to be between the two experiments.

Fig. 10 shows the fluctuating pressure coefficients. These have been reported to vary significantly between individual windtunnel experiments. With the present simulations the coarse grid shows a lower fluctuating pressure coefficient on the top and bottom sides of

---

**Fig. 8.** The Reynolds stress terms in a stream-wise line through the cylinders center of gravity. 

(Lyn et al. 1995). Present simulations: --, Grid A; ---, Grid B; ----, Grid C. (a) is the $u'$ component; (b) is the $v'$ component; (c) is the cross correlation between $u'$ and $v'$.
As the grid is refined the fluctuating pressure coefficient increases. This is consistent with the increase in the fluctuating lift force coefficient during grid refinement (see Table 2).

Fig. 9. Time-mean surface pressure coefficient. -, (Bearman 1982); - - - - , (Lee 1975). Numerical: \( \times \), Grid A; \( \circ \), Grid B (each 2\textsuperscript{nd} point); \( \triangle \), Grid C (each 4\textsuperscript{th} point).

The finer grids show a variation of the fluctuating pressure coefficient that lies within the two experimental findings used for comparison here. For comparison the distribution reported by (Yu and Kareem 1997) has been included in the plot.

Fig. 10. Fluctuating surface pressure coefficients (rms of pressure fluctuation). - - - - , (Bearman and Obasaju 1982); - - - - , (Pocha 1971). Numerical: \( \times \), Grid A; \( \circ \), Grid B (each 2\textsuperscript{nd} point); \( \triangle \), Grid C (each 4\textsuperscript{th} point); - - - - - , (Yu and Kareem 1997).

The correlation of surface pressures is essential in the task of estimating global loads from local pressure distributions. It is due to the lack of full correlation that building codes provide smaller shape factors, and hence loads, for large surface areas, than for small areas. For instance
individual cladding elements on a building should be designed for a higher local surface pressure than the supporting frames that carry a large assembly of cladding elements and thus loads from a large area.

For the present simulations, the correlation of pressures in a cross section near the mid span has been investigated. A base point was chosen on the top surface 5/12 d downstream of the leading edge. The correlation of pressure between this point, and all other points in the cross section was then determined, and plotted in Fig. 11. The simulated correlations found with Grid A, appear to be in close agreement with the experimental findings.

**Fig. 11.** Circumferential correlation of surface pressures with respect to a pressure tab at the top surface, placed 5/12 d downstream from the front surface. —, (Lee 1975); *, Grid A.

### 5. CONCLUSION

The present test case has shown that the model can in fact simulate the flow around a square cylinder with results that are comparable or improved in comparison to the results of LES simulations. The SMoM model is thus found to provide an alternative to LES.

The SMoM model is still immature in comparison to LES models, and does still bear the flow dependent scale parameter \( b \). Thus, work is still need to address the general applicability of the SMoM model for civil engineering flows.

### REFERENCES


