MESHING STRATEGY EVALUATION FOR A SQUARE SHAPED BLUFF BODY UNDER HIGH REYNOLDS NUMBER CROSS FLOW

Filipe Branco Teixeira¹
Andrei Luís Garcia Santos²
Luiz Alberto Oliveira Rocha³
Liércio André Isoldi⁴
Elizaldo Domingues dos Santos⁵

Abstract: This work consists in the numerical evaluation of meshes employed in a two-dimensional, incompressible, and transient flow, with high Reynolds numbers and forced convection that passes through a square shaped bluff body. The objective of the study is to evaluate different strategies of mesh construction to solve this type of problem, seeking to reduce the computational cost and improve the solution. The simulations were performed for Reynolds and Prandtl numbers of $Re_0 = 22,000$ and $Pr = 0.71$. The turbulence behavior is solved with the RANS SST-$\kappa-\omega$ model. The evaluation of the solutions is performed analyzing the drag coefficient ($C_D$), Nusselt number ($Nu_D$) and Strouhal number ($St$). The use of an unstructured mesh with fully structured local refinement was able to validate the fluid dynamics of the case with a relative deviation of 14.95% for the velocity field, compared to the literature. The deviations for $C_D$ and $St$ were as low as 0.43% and 0.79% respectively. For thermal verification, the deviation of the mean local Nusselt number along the bluff body ($Nu_D$) was 5.93%, and the deviation for the global Nusselt number ($Nu_g$) was as low as 0.56% compared to the literature.

Keywords: High Reynolds number. Turbulence. Convection. Bluff body. CFD.

¹ Master in Oceanic Engineering, Universidade Federal do Rio Grande, e-mail: fbrancoteixeira@gmail.com.
² Undergrad in Mechanical Engineering, Universidade Federal do Rio Grande, e-mail: andrei_eng_mec@hotmail.com.
³ Doctor in Mechanical Engineering, Universidade Federal do Rio Grande do Sul, e-mail: luizrocha@mecanica.ufrgs.br.
⁴ Doctor in Mechanical Engineering, Universidade Federal do Rio Grande, e-mail: liercioisoldi@furg.br.
⁵ Doctor in Mechanical Engineering, Universidade Federal do Rio Grande, e-mail: elizaldodossantos@gmail.com.

Revista Mundi Engenharia, Tecnologia e Gestão. Paranaguá, PR, v.3, n.2, maio de 2018.
1 INTRODUCTION

Problems with fluid and thermal flows are found in several engineering areas. One of the most studied topics is precisely the phenomena that occur with tubes, bluff bodies and finned channels in cross flow (commonly found, for example, in heat exchangers). Currently, important studies have been carried out in the area of geometric evaluation of arrangements of cylinders and bluff bodies, seeking to improve the performance of the system according to multiple objectives. It has sought to find geometries that maximize the heat transfer (thermal function) and minimize the resistance to the flow (fluid dynamic function). The great majority of these studies analyze the behavior of laminar flows and, since the interest is the phenomenological study of processes such as advection and thermal diffusion, these flows tend to present more predictable and stable numerical results with good convergence and relative low computational cost.

However, most engineering problems with real application fall within the field of turbulent flow regimes (WILCOX, 2006). From the electronic component cooling system to automotive cooling systems and industrial thermal equipments, the flow regime employed is often turbulent. This is due to the need for a large thermal exchange in these equipments, also requiring large momentum of the refrigerant fluid.

The turbulent flows have a very complex physical nature and generally require sophisticated measuring instruments to be studied experimentally, which sometimes becomes economically unfeasible. Thus, few works are found in the literature related to the experimental study of turbulent flows on a bluff body. Of particular note are the studies by Igarashi (1986), Durao et. al. (1988) and Lyn et. al. (1995).

According to Wilcox (2006), the intricate mathematical modeling of turbulent flows, associated with the geometric and physical complexity of the problem, makes its analytical solution impossible. On the other hand, numerical methods have been presented in a technically and economically feasible way for researchers in the area due to the vertiginous development of high-speed computers with large storage capacity (MALISKA, 2004). Nevertheless, many
difficulties are encountered to evaluate this type of phenomenon, ranging from: the choice of numerical model; the implementation of appropriate boundary conditions; as well as the creation of a mesh that can correctly capture the velocity, temperature and pressure field gradients in the studied flow.

The simulation of turbulent flows over a single bluff body has been the subject of important recent studies such as that of Perng & Wu (2007) who used the Large Eddy Simulation (LES) approach to study the effects of natural convection and buoyancy on the bluff body. Subsequently, Ranjan & Dewan (2015) used a PANS (Partially-Averaged Navier-Stokes) model associated with the SST-\(\kappa-\omega\) turbulence modeling to compare two meshes, one structured with wall function and one unstructured with local refinement. Chen & Xia (2017) compared the RANS (Reynolds Averaged Navier-Stokes) Spalart-Allmaras and SST-\(\kappa-\omega\) models to solve the same problem. All these studies have great emphasis on mathematical and numerical modeling, but do not systematically analyze the generation of meshes that use diverse strategies in order to find patterns that reduce the computational cost while allowing studying geometries that are more complex.

Therefore, the present work intends to address the issue of the computational mesh for the solution of external and turbulent flows over bluff bodies, with a series of distinct meshing strategies that can, besides reducing the computational cost, lead to as close as possible results as those obtained in the literature, for the main flow parameters \((C_D, N Ud, \text{and } St)\). Although the phenomenon of turbulence is inherently three-dimensional, domains with two-dimensional meshes will be generated here, since the objective is to ascertain the different creation strategies. It is intended to use the recommendations obtained in this work for future studies of geometric evaluation on arrangements of bluff bodies.

2 MATHEMATICAL MODELING

The problem under study is a two-dimensional, incompressible high Reynolds number flow, in a transient regime, with forced convection and constant thermophysical properties. The domain employed is a two-dimensional
reconstruction of the domain used by Wiesche (2006), Ranjan & Dewan (2015) and Chen & Xia (2017) where the cartesian coordinate $z$ was suppressed and the dimension in $y$ was enlarged from $10D$ to $14D$. This was done in order to avoid numerical distortions originating from the symmetry boundary condition, since the former rapidly generated vortices that reached the upper and lower limits of the domain. The flow passes through a bluff square shaped body of edge $D$, positioned at the vertical center of the domain and at $9D$ of its entrance, as can be seen in Fig. 1, and is caused by the imposition of a constant velocity profile ($V_\infty$) at the domains inlet. The variables are treated so that is obtained $Pr = 0.71$ and $Re_D = 22,000$ for the Prandtl and Reynolds numbers respectively. The flow has also a prescribed temperature $T_\infty$ that is lower than the surface temperature of the bluff body $T_S$ by $30K$, so the forced convective heat transfer occurs due to this temperature difference. The other boundary conditions include the imposition of null heat flux and zero gauge pressure at the domains outlet, besides the condition of symmetry on the north and south faces.

**Figure 1** – Computational domain used on the simulations.

Source: Elaborated by the authors.

The modeling chosen for this study is based on the Reynolds Averaged Navier-Stokes (RANS) conservative equations, altogether with the equations of the SST-$\kappa-\omega$ model to tackle with turbulence closure. It is known that turbulence is an inherently three-dimensional phenomenon, but the two-dimensional study aims at generating recommendations while reducing computational cost. The SST-$\kappa-\omega$ turbulence model is a two-equation model proposed by Menter (1993) as an alternative to the original $\kappa-\omega$ model that has too much sensitivity in the...
free-current regions. The SST (Shear Stress Transport) solves this problem, since it changes the behavior of the model resembling the formulation $\kappa-\varepsilon$ in the free flow. It is worth mentioning that the analysis was performed when the flow reached the steady state regime (when the turbulence structures started to present repetitions in time). However, the simulations are performed in transient regime, since turbulent flows are naturally time dependent.

2.1 Conservative equations and turbulence model

The mean time-averaged conservative equations for mass, momentum in $x$ and $y$ and energy respectively are given as seen in Bejan (2004):

\[
\frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} = 0
\]

(1)

\[
\frac{\partial (\rho \bar{u})}{\partial t} + \frac{\partial (\rho \bar{u} \bar{u})}{\partial x} + \frac{\partial (\rho \bar{u} \bar{v})}{\partial y} = -\frac{\partial \bar{P}}{\partial x} + (\mu + \mu_t) \left( \frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2} \right)
\]

(2)

\[
\frac{\partial (\rho \bar{v})}{\partial t} + \frac{\partial (\rho \bar{u} \bar{v})}{\partial x} + \frac{\partial (\rho \bar{v} \bar{v})}{\partial y} = -\frac{\partial \bar{P}}{\partial y} + (\mu + \mu_t) \left( \frac{\partial^2 \bar{v}}{\partial x^2} + \frac{\partial^2 \bar{v}}{\partial y^2} \right)
\]

(3)

\[
\frac{\partial \bar{T}}{\partial t} + \frac{\partial (\bar{u} \bar{T})}{\partial x} + \frac{\partial (\bar{v} \bar{T})}{\partial y} = (\alpha + \alpha_t) \left( \frac{\partial^2 \bar{T}}{\partial x^2} + \frac{\partial^2 \bar{T}}{\partial y^2} \right) + q'''
\]

(4)

where: $\rho$ is the density (kg/m³), $x$ is the x-axis cartesian coordinate (m); $u$ is the velocity component in the $x$-axis direction (m/s), $y$ is the $y$-axis cartesian coordinate (m); $v$ is the velocity component in the $y$-axis direction (m/s); $P$ is the pressure (N/m²); $T$ is the temperature (K); $c_p$ is the specific heat at constant pressure (J/kg.K) and $q'''$ is the energy source (W/m³), that in this case is zero.

To solve the turbulence problem, the model chosen is the SST-$\kappa-\omega$ initially proposed by Menter (1993). It is a model of two equations derived from
the original $\kappa-\omega$ formulation presented by Wilcox (1988). Its major advantage is that it uses a blending function, which keeps the $\kappa-\omega$ formulation in the inner parts of the boundary layer close to the wall, changing to a $\kappa-\varepsilon$ behavior as it reaches the free-stream thus reducing the weaknesses of each model. In addition, this model modifies the definition of kinematic eddy viscosity to include the turbulence Shear Stress Transport (SST). Thus, the equations for the turbulent kinetic energy and for the specific dissipation rate are given by:

$$\frac{\partial k}{\partial t} + \frac{\partial (\bar{u} j k)}{\partial x_i} = \bar{p}_k - \frac{k^{3/2}}{L_T} + \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_k \mu_T) \frac{\partial k}{\partial x_i} \right]$$

(5)

$$\frac{\partial \omega}{\partial t} + \frac{\partial (\bar{u} i \omega)}{\partial x_i} = \left( \frac{\alpha}{\mu_T} \right) \bar{p}_k - \beta \omega^2 + \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_\omega \mu_T) \frac{\partial \omega}{\partial x_j} \right] +$$

$$+2(1 - F_1) \frac{\sigma_\omega \omega}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}$$

(6)

where: $k$ is the turbulent kinetic energy, $\omega$ is the specific dissipation rate, $\mu_T$ is the kinematic eddy viscosity, the constants $P_k$, $\beta$, $\alpha$, $\beta_1$, $\alpha_k$, $\sigma_k$, $\sigma_\omega$, $\sigma_2$, $\beta_2$, $\sigma_2$ and $\sigma_{\omega 2}$ are the same as seen in Menter (1994) and $F_1$ is a blending function between variables and constants defined by:

$$F_1 = \tanh \left\{ \min \left[ \max \left( \frac{\sqrt{k}}{\beta^* \omega y}, \frac{500 \nu}{\beta^* \omega y^2} \right), 4 \rho \sigma_{\omega 2} k \right] \right\}$$

(7)

3 NUMERIC MODELING

Gmsh, an open source meshing software was adopted for the construction of the computational meshes. Always using rectangular elements, the constructive complexity of the mesh is increased gradually for each simulated case. Six different mesh generation strategies were studied according to Table 1. It is worth mentioning that a larger total number of cells does not necessarily represent a more complex generation strategy, since one of the
The main intention of this work is to reduce the computational effort by using strategies that are more efficient.

The strategies used were as follows:

1. Structured mesh with horizontal refinement;
2. Structured mesh with progressive cross shaped refinement;
3. Structured mesh with fixed cross shaped refinement;
4. Structured mesh with partially progressive cross shaped refinement;
5. Unstructured mesh with totally structured local refinement;
6. Unstructured mesh with partially structured local refinement

Table 1 – Cell count in each domain line.

| Line          | Nº of cells |
|---------------|-------------|
|               | Strategy 1  | Strategy 2 | Strategy 3 | Strategy 4 | Strategy 5 | Strategy 6 |
| Inlet and Outlet | 110         | 180        | 500        | 560        | 160        | 160        |
| Symmetries    | 470         | 280        | 770        | 610        | 80         | 80         |
| Body faces    | 20          | 50         | 60         | 200        | 200        | 200        |
| Auxiliary 1   | -           | -          | -          | -          | 60         | 70         |
| Auxiliary 2   | -           | -          | -          | -          | 60         | 30         |
| Total approx. | 51,000      | 48,000     | 386,000    | 301,000    | 107,500    | 107,500    |

**Source:** Elaborated by the authors.

Figures 2 (a) to (d) illustrate a schematic cross-section of the different structured mesh strategies employed. While Figures 3 (a) and (b) present the unstructured mesh strategies used in this study, detailing the local refinement in the wall region.

For the processing, Eqs. (1) to (7) are solved through the ANSYS FLUENT® computational fluid dynamics software, which is based on the Finite Volume Method (FVM) (VERSTEEG & MALALASEKERA, 2007; PATANKAR, 1980; ANSYS, 2017). The solver is pressure based and the simulation runs on a transient regime. The turbulence model used is the SST $\kappa-\omega$ adjusted with the program default values and the pressure-velocity coupling is performed using the SIMPLE-C (Semi-Implicit Method for Pressure Linked Equations-Consistent) algorithm.
For the advective terms treatment such as the conservative equations of momentum; energy; turbulent kinetic energy and specific dissipation rate, a second order Upwind interpolation scheme is used, while for the pressure the second order scheme is employed. The maximum allowed values for the residues between two iterations are $1 \times 10^{-5}$ for the mass, $1 \times 10^{-6}$ for the momentum; turbulent kinetic energy and specific dissipation rate and $1 \times 10^{-8}$ for the energy.

The simulations are carried out in two stages, the first consists in the attainment of the flow stability, at which point the fluid and dynamic behavior presents a pattern with repetitions in time. After this pattern is reached, the second stage of the simulation is performed with the time statistics sampling option activated, thus collecting the statistics of the turbulence with each time step. This allows the analysis of the average fields of temperature, speed, pressure among others already with the stabilized flow, thus avoiding using data of when the flow had not yet reached the obstacle.
All simulations were performed using a computer with an Intel Core i7 5820K 3.3 GHz hexa-core processor and 16GB of RAM. The approximate processing time for the simulations varied between 11 and 61 hours, according to the complexity of the mesh used.

### 4 RESULTS AND DISCUSSION

The simulations for the six proposed cases were performed and compared with each other and with the literature. Numerous parameters (field of velocities, mean local Nusselt number, \(Nu_D\), global Nusselt number \(Nu_g\), drag coefficient \(C_D\) and Strouhal number \(St_f\)) were used to determine the level of refinement and the strategy with the highest potential to be used with more complex geometries and at the same time keep the computational requirements at levels not prohibitive for future researches. It is worth remembering that the purpose of this work is to evaluate the mesh construction strategy and not the...
simple complexity or number of computational cells. Therefore, we tried to make a fair comparison respecting the different size of meshes, but analyzing the behavior of the solution (efficiency between a better result and the less time needed to reach it with the same computational resources).

Figure 4 shows the solution for one of the evaluated parameters, the domains centerline mean velocity. In order to quantify a relative deviation, the results are compared with the experiments of Lyn et. al. (1995).

As can be seen in Fig. 4 (a), the mesh strategies 1 and 2 present a solution quite distant from the others, which could already be expected in view of the much lower number of computational cells in the meshes (51,000 and 48,000 respectively) especially in the wall region. However, case 2 shows a significant improvement (48.81% versus 75.07% relative deviation of case 1), even with the reduction of the mesh size, only by the change in the constructive strategy, that is, the cross-shaped refinement.

![Figure 4 – Centerline mean velocity: (a) Studied cases, (b) Comparison of the mesh strategy 5 with the literature.](image)

Source: Elaborated by the authors.

Although with some changes, cases 3 and 4 also used cross-shaped refinements, but with considerably more refined meshes. These cases obtained, for the parameter in question, deviations of 28.25% and 9.69%. However, as can be seen by the simulation time required in Table 2, there was a big increase, on the computational effort. Finally, cases 5 and 6, through unstructured meshes that are substantially smaller than those of cases 3 and 4,
were able to reach deviations of 14.85% and 17.04%, respectively (tolerable values for this kind of flows over bluff bodies). Figure 4 (b) shows the graphical comparison of case 5 with solutions from the literature, since this strategy, besides achieving acceptable deviation, reduced the simulation time by approximately 40% against the best case (strategy 4) and has the capacity to adapt to more complex geometries. It is worth to notice that cross-shaped mesh constructions are practically impossible to employ on the presence of not aligned bluff bodies for example.

Figure 5 shows the time-averaged mean local Nusselt number \( (Nu_D) \) as a function of the position of the bluff body for the flow at \( Re_D = 22,000 \) and \( Pr = 0.71 \). Figure 5(a) shows the solutions for the local Nusselt number around the bluff body for the six cases studied. For this parameter, it is noted that the main factor for a better solution is the refinement directly applied on the wall region. This is verified through cases 4, 5 and 6, all with two hundred cells in each face of the obstacle. Here the relative deviation quantification is performed comparing to Ranjan & Dewan (2015). The relative deviations of the abovementioned cases were 9.86%, 5.93% and 4.34% respectively.

**Figure 5** – Local Nusselt number: (a) Studied cases, (b) Comparison of the mesh strategy 5 with the literature.

![Graphs showing local Nusselt number](source: Elaborated by the authors.)

Figure 5 (b) again presents the comparison of case 5 versus the literature, since although it showed slightly greater deviation than case 6, its simulation time was approximately 10% lower, giving a considerable advantage.
considering that the two are in the acceptable range of deviation for problems with high Reynolds numbers.

Finally, Figs. 6 (a) and (b) illustrate for each case, the instantaneous $y^+$ profiles as a function of the bluff body position, an important parameter for checking the refinement state of the mesh in the wall region. According to Wilcox (2006), turbulence models $\kappa-\omega$ require that this value be of $y^+ \leq 1$. In this way, only the unstructured mesh cases with local refinement presented mean $y^+$ less than 1, being 0.90 for case 5 and 0.67 for case 6.

**Figure 6** – Instantaneous $y^+$ profile: (a) Low wall refinement meshes, (b) High wall refinement meshes.

![Graphs showing $y^+$ profiles for different mesh strategies](image)

**Source:** Elaborated by the authors.

Table 2 presents a general summary of the six strategies studied and a comparison with numerical and experimental $Nu_g$, $C_D$ and $St$ results available in the literature. It is observed that the relative deviations from case 5 to the global Nusselt number ($Nu_g$), drag coefficient ($C_D$) and Strouhal number ($St$), are as low as 0.56%, 0.43% and 10.71% respectively when compared to the experiments of Igarashi (1985). The comparison with Lyn et. al. (1995) is also satisfactory, with deviations of 9.52% for the $C_D$ and 0.79% for the $St$. In comparison with numerical works, case 5 presents an average relative deviation of 5.50% for the $C_D$ and 6.72% for the $St$ compared to Bouris & Bergeles (1999), as well as 5.42% for the $Nu_g$ compared to Chen & Xia (2017).
Table 2 – Results comparison with literature.

| Source                        | Parameters | Sim. time (h) | Method       |
|-------------------------------|------------|---------------|--------------|
|                               | $N_u_g$    | $C_D$         | $St$         |               |
| Hilpert (1933)$^1$            | 115.8      | -             | -            | Correlation   |
| Sparrow et. al. (2004)$^1$    | 162.5      | -             | -            | Correlation   |
| Igarashi (1985)$^2$           | 107.6      | 2.31          | 0.14         | Experimental  |
| Durao (1988)$^2$              | -          | -             | 0.133        | Experimental  |
| Franke & Rodi (1991)          | 162.5      | -             | -            | Experimental  |
| Lyn et. al. (1995)            | 156.2      | -             | 0.19         | LES           |
| Bouris & Bergeles (1999)      | 100.4      | 1.97          | 0.129        | PANS SST $\kappa-\omega$ |
| Wiesche (2006)                | 101.5      | -             | 0.134        | SST - LES     |
| Ranjan & Dewan (2015)         | 54.7       | -             | 12           | SST $\kappa-\omega$ |
| Strategy 1                    | 81.4       | 2.40          | 0.131        | SST $\kappa-\omega$ |
| Strategy 2                    | 102.1      | 2.22          | 0.137        | SST $\kappa-\omega$ |
| Strategy 3                    | 114.4      | 2.24          | 1.21         | SST $\kappa-\omega$ |
| Strategy 4                    | 107.0      | 2.30          | 0.125        | SST $\kappa-\omega$ |
| Strategy 5                    | 106.7      | 2.28          | 0.127        | SST $\kappa-\omega$ |

$^1$Values calculated from experimental acquired coefficients.  
$^2$The measurement uncertainty is not mentioned on the work.  
Source: Elaborated by the authors.

5 CONCLUSIONS

A numerical study was carried out to evaluate a series of different mesh generation strategies in a high Reynolds number flow with forced convection that passes through a square shaped bluff body of dimension $D$. Six cases were simulated and their results compared to the literature. The simulations were performed, with constant Reynolds and Prandtl numbers for all cases ($Re = 22,000; Pr = 0.71$).

The proposed meshes were generated with the aid of open source code software Gmsh. For the processing phase, the two-dimensional dual-precision mode of the ANSYS FLUENT® software was used. The turbulence model employed was the SST $\kappa-\omega$. The pressure-velocity coupling is solved with the SIMPLE-C algorithm and the discretization of the fields of velocity, energy, turbulent kinetic energy and the specific dissipation rate by the second order Upwind scheme. The pressure is resolved with the second order scheme.

Revista Mundi Engenharia, Tecnologia e Gestão. Paranaguá, PR, v.3, n.2, maio de 2018.
The results showed that the mathematical and numerical modeling used could accurately represent the time-averaged parameters of turbulent flows such as the Strouhal number, drag coefficient and Nusselt number even in two-dimensional domains. However, it is necessary to employ well-constructed meshes, sufficiently refined in the wall regions.

Case 5 presented a time-averaged local mean deviation within the tolerable range with 14.85% for the field of velocities in the fluid dynamic validation, compared to Lyn et.al. (1995) and 5.93% for \( \text{Nu}_D \) in the thermal verification, compared to Ranjan & Dewan (2015). Compared with the literature, the time-averaged global mean deviation for \( \text{Nu}_g \), \( C_D \) and \( St \) were as low as 0.56%, 0.43% and 0.79% respectively. This case is the more indicated (although it is not the one that leads to the smaller deviations in comparison with the literature) due to its greater flexibility to the assembly of the meshes, which allows its application in more complex geometries like arrangement of obstacles. The cross-shaped mesh of case 4, in spite of the excellent results obtained, is very complex for the assembly of misaligned bluff bodies, which makes difficult its future application in geometric evaluation studies. In future works, it is intended to use the strategy of case 5 to study the effect of geometry in complex arrangements of bluff bodies and on fluid dynamics and thermal performance subject to forced convective turbulent flows.

6. ACKNOWLEDGEMENTS

The authors F. B. Teixeira and A. L. G. Santos thanks CAPES for the master’s scholarship. The authors E. D. dos Santos, L. A. Isoldi and L. A. O. Rocha thank CNPq for the productivity research grant and financial support.

REFERENCES

ANSYS. Academic Version 18.0 (2017), – FLUENT User's Guide, ANSYS Inc.

Bejan, A. (2004), Convection Heat Transfer, John Wiley, Durham, USA.

Revista Mundi Engenharia, Tecnologia e Gestão. Paranaguá, PR, v.3, n.2, maio de 2018.
Bouris, D. & Bergeles, G. (1999), *2D LES of vortex shedding from a square cylinder*. Journal of Wind Engineering and Industrial Aerodynamics, v. 80, n. 1-2, p. 31-46. (DOI: 10.1016/S0167-6105(98)00200-1)

Chen, X. & Xia, H. (2017), *A hybrid LES-RANS study on square cylinder unsteady heat transfer*. International Journal of Heat and Mass Transfer, v.108, Part A, p. 1237-1254. (DOI: 10.1016/j.ijheatmasstransfer.2016.10.081)

Durao, D. F. G.; Heitor, M. V. and Pereira, J. C. F. (1988), *Measurements of turbulent and periodic flows around a square cross-section cylinder*. Experiments in Fluids. v. 6, n. 5, p. 298-304. (DOI: 10.1007/BF00538820)

Franke, R. & Rodi, W. (1991), *Calculation of vortex shedding past a square cylinder with various turbulence models*, Proceedings of 8th Symposium on Turbulent Shear Flows, Tech. Univ. Munich, Springer Berlin, p. 189-204.

Hilpert, R. (1933), *Wärmeabgabe von beheizten Drähten und Rohren im Luftstrom*. Forsch. Ingenieurwesen n. 4, p. 215–224.

Igarashi, T. (1986), *Local heat transfer from a square prism to an air stream*. International Journal of Heat and Mass Transfer, v.29, n. 5, p. 777-784. (DOI: 10.1016/0017-9310(86)90129-8)

Lyn, D. A.; Einav, S.; Rodi, W. and Park, J.-H. (1995), *A laser-Doppler velocimetry study of ensemble-averaged characteristics of the turbulent near wake of a square cylinder*. Journal of Fluid Mechanics, v. 304, p. 285-319. (DOI: 10.1017/S0022112095004435)

Maliska, C.R. (2004), *Transferência de calor e mecânica dos fluidos computacional*, Rio de Janeiro: LTC – Livros Técnicos e Científicos Editora S.A., 2ª Ed.

Menter, F. R. (1993), *Zonal Two Equation κ–ω Turbulence Models For Aerodynamic Flows*. AIAA 24th Fluid Dynamics Conference 1993, AIAA 93-2906.
Menter, F. R. (1994), *Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications*. AIAA Journal, v.32, n. 8. (DOI: 10.2514/3.12149)

Patankar, S. V. *Numerical Heat Transfer and Fluid Flow*, McGraw-Hill, New York, USA, 1980.

Perng, S.W. & Wu, H.W. (2007), *Buoyancy-aided/opposed convection heat transfer for unsteady turbulent flow across a square cylinder in a vertical channel*. International Journal of Heat and Mass Transfer, v. 50, n. 19-20, p. 3701-3717. (DOI: 10.1016/j.ijheatmasstransfer.2007.02.026)

Ranjan, P. & Dewan, A. (2015), *Partially Averaged Navier Stokes simulation of turbulent heat transfer from a square cylinder*. International Journal of Heat and Mass Transfer, v.89, p. 251-266. (DOI: 10.1016/j.ijheatmasstransfer.2015.05.029)

Sparrow, E. M.; Abraham, J. P. and Tong, J. C. K. (2004), *Archival correlations for average heat transfer coefficients for non-circular cylinders and for spheres in cross-flow*. International Journal of Heat and Mass Transfer v. 47, p. 5285-5296. (DOI: 10.1016/j.ijheatmasstransfer.2004.06.024)

Versteeg, H. K. & Malalasekera, W. (2007), *An Introduction to Computational Fluid Dynamics – The Finite Volume Method*, 1st Ed, Longman, England.

Wiesche, S. a.d. (2006), *Large-eddy simulation study of an air flow past a heated square cylinder*. Heat and Mass Transfer, v. 43, n. 6, p. 515-525. (DOI: 10.1007/s00231-006-0122-x)

Wilcox, D. C., *Turbulence Modeling for CFD*, third ed., DCW Industries, 2006.

Edição especial - XX ENMC (Encontro Nacional de Modelagem Computacional) e VIII ECTM (Encontro de Ciência e Tecnologia dos Materiais), realizado entre 16 e 19 de outubro de 2017 na cidade de Nova Friburgo – RJ.

Editor – Mateus das Neves Gomes

Revista Mundi Engenharia, Tecnologia e Gestão. Paranaguá, PR, v.3, n.2, maio de 2018.