GRID CONVERGENCE STUDY OF A CYCLONE SEPARATOR USING DIFFERENT MESH STRUCTURES

Article Highlights
- A numerical study of a cyclone separator was carried out using CFD techniques
- A grid independence analysis was carried out to optimize simulation effort and accuracy
- Two mesh structures were used to find out the best numerical mesh for these simulations
- Wall-refined meshes better represent collection efficiency
- Regular element meshes better represent pressure drop

Abstract
In a cyclone design, pressure drop and collection efficiency are two important performance parameters to estimate its implementation viability. The optimum design provides higher efficiencies and lower pressure drops. In this paper, a grid independence study was performed to determine the most appropriate mesh to simulate the two-phase flow in a Stairmand cyclone. Computational fluid dynamic (CFD) tools were used to simulate the flow in an Eulerian-Lagrangian approach. Two different mesh structure, one with wall-refinement and the other with regular elements, and several mesh sizes were tested. The grid convergence index (GCI) method was applied to evaluate the result independence. The CFD model results were compared with empirical correlations from bibliography, showing good agreement. The wall-refined mesh with 287 thousand elements obtained errors of 9.8% for collection efficiency and 14.2% for pressure drop, while the same mesh, with regular elements, obtained errors of 8.7% for collection efficiency and 0.01% for pressure drop.

Keywords: CFD, cyclone separator, bagasse soot, numerical mesh, grid convergence index.

As researchers came into a consensus about all harmful effects of fossil fuels on health and due to the high prices of gasoline, renewable fuels such as ethanol from sugarcane became a solution that is not only cheaper, but also less harmful to health and of lower impact to the environment.

As a secondary product, the bagasse of the milled sugarcane still has energetic value and it is generally combusted, in order to produce electric energy to sustain the whole industry and, sometimes, sell the exceeding energy. However, its combustion produces sugarcane bagasse soot, a particulate matter also noxious to health and environment that cannot be simply dumped at the environment.

So, in order to control the emission of particulate matter at atmosphere, industries install specific equipment for gas effluent treatment at the outlet of the boiler, such as cyclone separators, electrostatic precipitators or Venturi scrubbers. One of the main used device on air treatment is the cyclone separator, due to its low cost on building, maintenance and capacity of operating at high pressures and temperatures [1,2], and its wide range of gas flow treatment, reaching values between 50 and 50000 m³/h [3].

Even though these advantages, its usage is limited to its collection efficiency of particles with diameter lower than 2 µm, compared to other equipment.
so that cyclone separators are generally used as a pre-collection device, attached to other equipment with higher efficiency.

The usage of computational fluid dynamics (CFD) is a cheap and reliable way to predict the complex flow field and particle trajectories inside a cyclone separator [5,6] in order to optimize the viability of implementation of equipment in a process, saving time and money for the industry. In order to optimize a simulation, improving its accuracy without increasing the simulation time and machine effort, a grid independence study should be performed before running cases.

When studying cyclone separator by CFD methods, researchers emphasize on optimization of the device, by analyzing the effects of the dimensions and operational conditions. Zhao, Su and Zhang [7] compared the effects of a spiral double inlet and a single inlet in the performance, obtaining a better gas flow pattern in a double spiral inlet, while Elsayed and Lacor [8] studied the performance of the device comparing different widths and heights, concluding that the width is more relevant than the height for collection efficiency.

Studies of effects of the height of the cyclone are carried out for different cone and cylinder lengths. It is common to see in the literature studies of the cone effects, comparing the performance of different heights and bottom diameter [9-11], obtaining better collection efficiency as the bottom diameter is reduced. In this research field, Brar, Sharma and Elsayed [12] carried out numerical simulations not only for different cone lengths, but also for different cylinder heights, concluding that increasing the cylinder mainly saves more pressure drop, while the major benefit of increasing the cone is the improvement of the collection efficiency.

It is recurring to see studies that optimize the performance by restructuring the depth and diameter of the vortex finder. Recent researches showed that a smaller pressure drop per flow rate unit can be achieved when a cone-shaped vortex finder is used, although the collection efficiency was insignificant for smaller vortex finder [13,14]. Other vortex finder innovation is seen in the work of Safikhani and Mehrabia [15], whose proposal consists of an outer cylinder and vortex limiter, instead of a conical part, decreasing the pressure drop.

The mesh generation is one of the most important requirements in CFD simulation. Understanding the influence of the mesh size and structure on the flow fields is essential to get reliable results. The analysis of different mesh parameters allows the obtaining of a suitable mesh for a specific problem with acceptable computational effort and numerical accuracy [16]. Barthe and Zhang [17], for example, showed the importance of adapting the fluid mesh when high gradients are present or boundary layer effects are important and proposed a mesh adaptivity procedure using triangular and tetrahedral elements in the mesh.

It is known that great velocity and pressure gradients are observed in cyclone separators, so that wall effects are essential to be analyzed in CFD simulations of these devices. It is common to see meshes with a high quantity of elements, in order to obtain accurate results near walls [8,15,17]. However, few studies are concerned with the influence of the mesh structure on the flow field in cyclones. In this context, the present study aims to propose proper mesh structure and size for a laboratory Stairmand cyclone separator with low quantity of elements without losing accuracy. For this purpose, a grid independence study was performed, using the grid convergence index (GCI) method, which consists in quantifying the numerical uncertainty by basing on the Richardson extrapolation method for discretization error estimation, so that a consistent and reliable report of the grid refinement and its accuracy is obtained [18].

**METHODOLOGY**

**Design and meshes generation**

The simulated cyclone has a rectangular entry, so, it has seven essential geometry parameters sized in function of its body diameter, $D_c$, as shown in Figure 1. The inlet has a height, $a$, and width, $b$. The outlet of particle matter has a diameter $B$, while the outlet of air is a cylinder with diameter $D_2$ and a depth of $S$. The high of the device is divided in $h_c$, corresponding to the cylindrical part, and $H_c$, corresponding to its total high.

The cyclone simulated was based on a high efficiency Stairmand model, with ratios and dimensions given in Table 1.

So, the computational domain was generated in software Design Modeler, available on ANSYS 14.5 packages. The complex geometry of this cyclone was subdivided in 33 simple bodies, such as cones and cylinders divided in four parts, in order to match with the rectangle that guides the O-grid generated, as seen in Figure 2.

Using this method, generation of well-structured meshes got simpler when using software Meshing, also available on ANSYS 14.5 packages, with elements exclusively hexahedral, so the simulation could become easier to converge, reducing its process time and machine efforts, was favored.

In order to apply the GCI method, 12 well-structured meshes with a refinement ratio between 1.3 and
1.33 were generated. They were divided in two groups according to the uniformity of the mesh elements - 6 with regular elements and 6 with the same number of elements of the meshes with regular elements, but with wall refinement with a bias ratio of 5. To estimate discretization error, both collection efficiency and pressure drop were used.

Figure 1. Scheme of a cyclone separator geometry parameters. Source: adapted from Wang (2004).


<table>
<thead>
<tr>
<th>Parameter</th>
<th>a</th>
<th>b</th>
<th>B</th>
<th>Dc</th>
<th>Dh</th>
<th>h2</th>
<th>H1</th>
<th>S</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dimension, m</td>
<td>0.12</td>
<td>0.05</td>
<td>0.09</td>
<td>0.23</td>
<td>0.12</td>
<td>0.35</td>
<td>0.93</td>
<td>0.12</td>
</tr>
<tr>
<td>Ratio</td>
<td>0.5</td>
<td>0.2</td>
<td>0.375</td>
<td>1</td>
<td>0.5</td>
<td>1.5</td>
<td>4</td>
<td>0.5</td>
</tr>
</tbody>
</table>

Mathematical model

As a device operating in high velocities, they present complex flow profiles. Two of the main turbulence models suitable to this simulation, also available in commercial software packages are the Reynolds stress model (RSM) and the large Eddy simulation (LES) [19,20]. Due to the complexity of LES model and computational efforts required by it, RSM is a common choice to describe the developed flow. By using the RSM, the exact transport equation is given by [21]:

$$\frac{\partial}{\partial t} \left( \rho \mathbf{u}_i \mathbf{u}_j \right) + \frac{\partial}{\partial x_k} \left( \rho \mathbf{u}_k \mathbf{u}_i \mathbf{u}_j \right) = D_i + P_i + \Pi_i + \epsilon_i$$  \hspace{1cm} (1)

where the first term is the local time derivative of stress, the second term corresponds to the convective transport, the stress diffusion term, $D_i$, is given by:

$$D_i = - \frac{\partial}{\partial x_k} \left( \rho \mathbf{u}_i \mathbf{u}_j \mathbf{u}_k \right) + (\rho \mathbf{u}_i \mathbf{u}_j) \delta_k + (\rho u_i u_j) \delta_k - \mu \left( \frac{\partial}{\partial x_k} \mathbf{u}_i \mathbf{u}_j \right)$$  \hspace{1cm} (2)

The shear production, $P_i$, is given by:

$$P_i = -\rho \left( \mathbf{u}_i \mathbf{u}_j \mathbf{u}_k - \mathbf{u}_j \mathbf{u}_k \mathbf{u}_i \right)$$  \hspace{1cm} (3)

The pressure-strain, $\Pi_i$, is given by:

$$\Pi_i = \rho \left( \frac{\partial \mathbf{u}_i}{\partial x_j} + \frac{\partial \mathbf{u}_j}{\partial x_i} \right)$$  \hspace{1cm} (4)

The dissipation term, $\epsilon_i$, given by:

$$\epsilon_i = -2\mu \left( \frac{\partial \mathbf{u}_i}{\partial x_k} \right) \frac{\partial \mathbf{u}_j}{\partial x_k}$$  \hspace{1cm} (5)

By using the one-way coupling, the discrete phase follows a fixed continuous phase flow field, in which the effects of the discrete phase on the continuum and the interaction between particles are neglected. The effects of gravity and gas drag force describe the flow, and the momentum equation of a particle is described as:

$$\frac{du}{dt} = F_o \left( \mathbf{u} + \mathbf{u}' - u_p \right) - g$$  \hspace{1cm} (6)

$$\frac{dv}{dt} = F_o \left( \mathbf{v} + \mathbf{v}' - \mathbf{v}_p \right) + \frac{w_p^2}{f_p}$$  \hspace{1cm} (7)
\[
\frac{dw_p}{dt} = F_D (\overrightarrow{w} + \overrightarrow{w_p} - \overrightarrow{w_p})/r_p
\]  
(8)

where the momentum transport coefficient between fluid and particles, \( F_D \), is given by:
\[
F_D = \frac{18 \mu C_D}{\rho_p d_p^2} 24
\]  
(9)

the drag coefficient for particles, \( C_D \), is given by:
\[
C_D = \begin{cases} 
\frac{24}{\text{Re}_p} & \text{Re}_p \leq 1 \\
\frac{24(1 + 0.15 \text{Re}_p^{0.687})}{\text{Re}_p} & 1 < \text{Re}_p \leq 1000 \\
0.44 \frac{\text{Re}_p}{\text{Re}_p > 1000} & \text{Re}_p > 1000
\end{cases}
\]  
(10a)  
(10b)  
(10c)

and Re<sub>p</sub> is the particle Reynolds number, given by:
\[
\text{Re}_p = \frac{\rho_p d_p |\overrightarrow{u}_p - \overrightarrow{v}_p|}{\mu}
\]  
(11)

where \( \overrightarrow{v} \) assumes velocity components \( u, v \) and \( w \).

**Case set up**

A gas-solid mixture at 20 m/s and 45.5 °C of air, with density of 1.103 kg/m³ and viscosity of 1.8582×10⁻⁵ Pa·s, and sugarcane bagasse soot with average particle diameter of 9 µm and density of 2351.1 kg/m³.

As the focus of this paper is to analyze the structure of the meshes, the diameter of the particle was set as constant, in order to simplify the simulations, which were carried out using Fluent Solver 14.5 in an Eulerian-Lagrangian approach, in which the gas was treated as continuum phase, particulate phase was treated as single particles and particle trajectories, representing a stream of particles, are calculated as a result of a balance of forces acting on them, by using a one-way coupling.

Each simulation was set up in a transient state with a time step of 0.0001 second for 10000 time steps, in order to simulate a cyclone operating for 1 s. This was enough to ensure that the inlet static pressure converged to a steady state.

Spatial discretization was carried out in a second order upwind Scheme, adopting the spherical drag law model for drag coefficient and the SIMPLE scheme for pressure-velocity coupling.

The solution was set to converge when the residuals of all the flow variables, such as continuity, turbulent kinetic energy, turbulent dissipation and stress tensors, fell below 1×10⁻⁵ and the monitored total pressures at the surface of inlet and the surface of the outlet of air became steady. All the numerical simulations were carried out using the default relaxation factor values in the Fluent solver.

As soon as static pressure of air at the inlet converged to a steady value, particles were injected in the domain at 20 m/s using a one-way coupling scheme, in which the effects of interaction between particles and the gas are neglected, and its flow was tracked, using the discrete phase model (DPM). In this model, particles were set up to reflect on wall, escape on air outlet and get trapped when reaches the bottom of the cyclone.

In order to use this coupling scheme, it is necessary to assume that the dispersed phase occupies a low volume fraction, generally lower than 10% [18]. To ensure that this condition was suited, the injection of particles was set up to be proportional to the number of elements in the inlet of the numerical mesh.

**Grid independence analysis**

In this study, the GCI method was used to optimize the simulation, choosing a numerical mesh which results in higher precision and lower required machine effort.

The procedure for estimation of discretization error suggested [20] is, first of all, defining a representative mesh size \( h \), given by:
\[
h = \left[ \frac{1}{N} \sum_{i=1}^{N} \Delta V \right]^{1/3}
\]  
(12)

where \( \Delta V \) is the volume of the \( i \) cell and \( N \) is the total number of cells used for the computations.

It is desirable a grid refinement factor, given by Eq. (13), greater than 1.3 and constant, based on experimental studies [20]:
\[
r = \frac{h_{\text{coarse}}}{h_{\text{fine}}}
\]  
(13)

After selecting the 3 finest meshes and running simulations to determine the values of relevant variables, \( \varphi \), the apparent order, \( p \), is calculated using fixed-point iteration in the equations:
\[
p = \frac{1}{\ln r_2} \ln \frac{r_2}{r_2^{p_2}} + q(p)
\]  
(14)

\[
q(p) = \ln \left( \frac{r_2^p - s}{r_2^{p_2} - s} \right)
\]  
(15)

\[
s = 1 \cdot \text{sign} \left( \frac{r_2^p - s}{r_2^{p_2} - s} \right)
\]  
(16)
where \( h_1 < h_2 < h_3 \), \( r_{21} = h_2/h_1 \), \( r_{32} = h_3/h_2 \), \( \epsilon_{32} = \phi_3 - \phi_2 \),
\( \epsilon_{21} = \phi_2 - \phi_1 \) and \( \phi_k \) is the solution on the \( k \)th mesh.

So, extrapolated values can be calculated by the Eq. (16), and similarly it can be done for \( \epsilon_{k \text{ext}} \). By calculating the extrapolated values, approximate relative error and extrapolated relative error can be estimated by Eqs. (17) and (18), respectively:

\[
\frac{\phi_{21}^{\text{ext}}}{\phi_1} = \frac{r_{21}^2 \phi_2 - \phi_1}{r_{21}^2 - 1} \tag{17}
\]

\[
\frac{\epsilon_{k \text{ext}}^{\text{21}}}{\epsilon_1} = \frac{\phi_{k \text{ext}} - \phi_1}{\phi_{k \text{ext}} - \phi_1} \tag{18}
\]

\[
\frac{\epsilon_{k \text{ext}}^{\text{21}}}{\epsilon_1} = \frac{\phi_{k \text{ext}} - \phi_1}{\phi_{k \text{ext}} - \phi_1} \tag{19}
\]

At last, the grid convergence index can be calculated by the Eq. (19) as follows:

\[
GCI_{\text{ext}}^{21} = \frac{1.25 \epsilon_{k \text{ext}}^{\text{21}}}{\epsilon_1} \tag{20}
\]

RESULTS AND DISCUSSION

Following GCI criteria and based on an initial coarse mesh with 10590 elements, the 6 different sizes of numerical meshes and the grid refinement factor were obtained, as shown in Table 2. As the sizes of the numerical meshes were established, two different kinds of meshes were generated for each size. The first one had only regular elements, while the other had wall-refined elements. The meshes quality statistics, shown in Table 3, in order to analyze its structures, shows that none of the 12 meshes had average aspect ratio over 6.1, average orthogonality below 0.97 or average skewness over 0.125.

In order to analyze the tangential and axial velocities of air and total pressure according to their radial position, an X-axis based line was set up in the middle of the cylindrical section of the device, obtaining profiles shown in Figure 3. The profiles were not only developed according to expectations, but also got better defined as meshes were refined.

Analyzing the figures above, each one has a trend line well developed at meshes with at least 129 thousand elements for wall-refined meshes and 287 thousand elements for regular element meshes. This difference can be easily explained due to the fact walls in a cyclone separator are the critical region. Thus, in order to obtain more accurate results with less computational efforts, a better refinement on walls is recommended.

In both situations, the ideal would be using meshes with 683 thousand elements, in which results got more precise. However, one of the objectives of this study was optimizing both accuracy and computational effort, so it was more viable to use the mesh with 287 thousand wall-refined elements.

The chosen mesh has around 287 thousand elements, all of them hexahedral, which 95.6% of them has orthogonal quality over 0.96 and 99.99% of them has an aspect ratio below 20. Thus, this mesh is considered well-structured, as seen in Figure 4, and has reliable results.

As soon as pressures at inlet, \( P_{\text{in}} \), and outlet, \( P_{\text{out}} \), converged to a steady value, pressure drop, \( \Delta P \), can be estimated by the equation:

\[
\Delta P_{\text{CFD}} = P_{\text{in}} - P_{\text{out}} \tag{21}
\]

while collection efficiency, in a CFD approach, can be

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Number of elements</th>
<th>( r )</th>
<th>( \Delta P_{\text{CFD}} ) / Pa</th>
<th>( \Delta P_{\text{correlation}} ) / Pa</th>
<th>( \delta % )</th>
<th>( \eta_{\text{CFD}} ) / %</th>
<th>( \eta_{\text{correlation}} ) / %</th>
<th>( \delta % )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10590</td>
<td>-</td>
<td>533.73</td>
<td>1104.14</td>
<td>51.66</td>
<td>80</td>
<td>70.44</td>
<td>41.57</td>
</tr>
<tr>
<td>2</td>
<td>24864</td>
<td>1.3291</td>
<td>817.54</td>
<td>1104.14</td>
<td>25.96</td>
<td>66.67</td>
<td>70.44</td>
<td>5.35</td>
</tr>
<tr>
<td>3</td>
<td>58230</td>
<td>1.328</td>
<td>1007.75</td>
<td>1104.14</td>
<td>8.73</td>
<td>75</td>
<td>70.44</td>
<td>6.47</td>
</tr>
<tr>
<td>4</td>
<td>129312</td>
<td>1.3047</td>
<td>1162.47</td>
<td>1104.14</td>
<td>5.28</td>
<td>73.75</td>
<td>70.44</td>
<td>4.70</td>
</tr>
<tr>
<td>5</td>
<td>287232</td>
<td>1.3048</td>
<td>1261.14</td>
<td>1104.14</td>
<td>14.22</td>
<td>77.35</td>
<td>70.44</td>
<td>9.81</td>
</tr>
<tr>
<td>6</td>
<td>683312</td>
<td>1.3349</td>
<td>1389.63</td>
<td>1104.14</td>
<td>25.86</td>
<td>76.68</td>
<td>70.44</td>
<td>8.72</td>
</tr>
</tbody>
</table>

Table 2. Comparison between CFD results in wall-refined and regular elements meshes and correlations expectations

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Number of elements</th>
<th>( r )</th>
<th>( \Delta P_{\text{CFD}} ) / Pa</th>
<th>( \Delta P_{\text{correlation}} ) / Pa</th>
<th>( \delta % )</th>
<th>( \eta_{\text{CFD}} ) / %</th>
<th>( \eta_{\text{correlation}} ) / %</th>
<th>( \delta % )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10590</td>
<td>-</td>
<td>270.81</td>
<td>1104.14</td>
<td>75.47</td>
<td>100</td>
<td>70.44</td>
<td>41.96</td>
</tr>
<tr>
<td>2</td>
<td>24864</td>
<td>1.3291</td>
<td>440.84</td>
<td>1104.14</td>
<td>60.08</td>
<td>83.33</td>
<td>70.44</td>
<td>18.30</td>
</tr>
<tr>
<td>3</td>
<td>58230</td>
<td>1.328</td>
<td>667.95</td>
<td>1104.07</td>
<td>39.51</td>
<td>75</td>
<td>70.44</td>
<td>6.47</td>
</tr>
<tr>
<td>4</td>
<td>129312</td>
<td>1.3047</td>
<td>830.31</td>
<td>1104.07</td>
<td>24.80</td>
<td>75</td>
<td>70.44</td>
<td>6.47</td>
</tr>
<tr>
<td>5</td>
<td>287232</td>
<td>1.3048</td>
<td>1104.07</td>
<td>1104.07</td>
<td>0.01</td>
<td>76.56</td>
<td>70.44</td>
<td>8.69</td>
</tr>
<tr>
<td>6</td>
<td>683312</td>
<td>1.3349</td>
<td>1194.17</td>
<td>1104.07</td>
<td>8.15</td>
<td>76.98</td>
<td>70.44</td>
<td>9.28</td>
</tr>
</tbody>
</table>
Table 3. Mesh quality parameters

<table>
<thead>
<tr>
<th>Mesh quality coefficient</th>
<th>Regular element meshes</th>
<th>Wall-refined meshes</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>10k</td>
<td>24k</td>
</tr>
<tr>
<td>Aspect Ratio</td>
<td>Minimum</td>
<td>1.302</td>
</tr>
<tr>
<td></td>
<td>Maximum</td>
<td>15.00</td>
</tr>
<tr>
<td>Orthogonality</td>
<td>Minimum</td>
<td>0.345</td>
</tr>
<tr>
<td></td>
<td>Maximum</td>
<td>1.00</td>
</tr>
<tr>
<td>Mean</td>
<td>0.976</td>
<td>0.981</td>
</tr>
<tr>
<td>σ</td>
<td>0.054</td>
<td>0.048</td>
</tr>
<tr>
<td>Skewness</td>
<td>Minimum</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>Maximum</td>
<td>0.789</td>
</tr>
<tr>
<td>Mean</td>
<td>0.121</td>
<td>0.099</td>
</tr>
<tr>
<td>σ</td>
<td>0.112</td>
<td>0.099</td>
</tr>
</tbody>
</table>

Figure 3. Flow profiles; dimensionless tangential velocity \((U/U_\text{in})\) for: a) wall-refined and b) regular element meshes; dimensionless axial velocity \((U_z/U_\text{in})\) for: c) wall-refined and d) regular element meshes; static pressure in a dimensionless radial position for: e) wall-refined and f) regular element meshes.
Figure 4. Views of the 287 thousand wall-refined elements mesh: a) superior view of the o-grid; b) interior view of the mesh.

The collection efficiency is estimated by the equation:

\[ \eta_{\text{CFD}} = \frac{n_{\text{trap}}}{n_{\text{inj}}} \]  

(22)

where \( n_{\text{trap}} \) is the number of tracked particles trapped in the bottom of the cyclone and \( n_{\text{inj}} \) is the number of tracked particles injected in the inlet of the cyclone separator.

To compare with CFD results, the following empirical correlations were used to estimate pressure drop:

\[ \Delta P_{\text{correlation}} = \frac{\rho v^2}{2} \Delta \bar{H} \]  

(23)

where \( \Delta \bar{H} \) is a dimensionless parameter that depends on the cyclone dimensions. One of the proposed a correlation to estimate \( \Delta \bar{H} \) is given by [22]:

\[ \Delta \bar{H} = 20 \left( \frac{a b}{D_c} \right) \left( \frac{S}{D_c} \sqrt{H/D_c} \frac{h_c}{D_c} \frac{B}{D_c} \right)^{\frac{1}{2}} \]  

(24)

For collection efficiency, the empirical correlations used to compare with CFD results are given by [4]:

\[ \eta_i = 1 - e^{-\frac{Q}{G}} \]  

(25)

where \( Q \) is the flow rate given in m³/h, \( G \) is a dimensionless geometry parameter, given by the equation:

\[ G = \frac{8K_c}{K_a^2 K_b^2} \]  

(26)

with:

\[ K_a = \frac{a}{D_c} \]  

(27)

\[ K_b = \frac{b}{D_c} \]  

(28)

\[ K_c = \frac{2V_s + V_{nl}}{D_c^3} \]  

(29)

where \( V_s \) is the annular volume between the central plane of the inlet duct and the bottom of the exit duct, \( S \), given by the equation:

\[ V_s = \pi \left( S - \frac{a^2}{2} \right) \left( D_c^2 - D^2 \right) \]  

(30)

and \( V_{nl} \) is an annular volume related to the vortex penetration inside the cyclone and can be calculated by the equation:

\[ V_{nl} = \frac{\pi D_c^2}{4} (h_c - S) + \left( \frac{\pi D_c^2}{4} \right) \left( \frac{Z_c + S - h_c}{3} \right) \left( 1 + \frac{d_c}{D_c} + \frac{d_c^2}{D_c^2} \right) - \frac{\pi D_c^2 Z_c}{4} \]  

(31)

where \( d_c \) is the diameter of the cyclone central axis, given by:

\[ d_c = D_c - (D_c - B) \left( \frac{S + Z_c - h_c}{H - h_c} \right) \]  

(32)

being \( Z_c \) the vortex natural length, given by:

\[ Z_c = 2.3 D_c \left( \frac{D_c^2}{ab} \right)^{\frac{1}{3}} \]  

(33)
and \( n \) is the vortex exponent, given by:

\[
n = 1 - \left[ 1 - 0.67 \left( \frac{D_c^{0.14}}{1000} \right) \right] \left( \frac{T}{283} \right)^{0.3}
\]

(34)

with \( D_c \) given in meters and the gas temperature \( T \) in Kelvin. Finally, the relaxation time, \( \tau \), given by:

\[
\tau = \frac{\rho D^2}{18 \mu}
\]

(35)

After calculating pressure drop and efficiency by CFD and empirical correlations, relative errors can be estimated by the equation:

\[
\delta = 100 \left( \frac{\phi_{\text{CFD}} - \phi_{\text{correlation}}}{\phi_{\text{correlation}}} \right)
\]

(36)

where \( \phi \) is the analyzed variable.

Another consequence of the results obtained on CFD model is the estimation of the apparent order and GCI for all collection efficiency and pressure drop for each mesh refinement, as shown in Table 4, where \( p \) is the apparent order for the \( P \) parameter \( \phi \), being 1 the collection efficiency and 2 the pressure drop.

Table 4. Apparent order and GCI for collection efficiency and pressure drop for different kinds of meshes

<table>
<thead>
<tr>
<th>Grid Step</th>
<th>Wall-refined meshes</th>
<th>Regular elements meshes</th>
</tr>
</thead>
<tbody>
<tr>
<td>GCI_{21}</td>
<td>5.753 6.04 0.6044</td>
<td>229.66 5.0741 7.73 4.3378 19.80</td>
</tr>
<tr>
<td>GCI_{32}</td>
<td>3.38 128.14</td>
<td>4.31 17.54</td>
</tr>
<tr>
<td>GCI_{43}</td>
<td>0.59 95.40</td>
<td>0.00 11.27</td>
</tr>
<tr>
<td>GCI_{54}</td>
<td>1.60 56.06</td>
<td>0.89 14.28</td>
</tr>
<tr>
<td>GCI_{65}</td>
<td>0.29 60.58</td>
<td>0.20 3.77</td>
</tr>
</tbody>
</table>

As seen in flow profiles on Figure 3, meshes with 683 thousand elements, for both regular elements and wall-refined meshes, better represented the trend of the expected flow, which could be confirmed by the GCI method.

Although the wall-refined mesh of 683 thousand elements developed a discretization error of 0.34% for pressure drop, a very low and good value, its discretization error for collection efficiency was 37.69%. So that, the better option was to simulate this device using the 287 thousand elements with wall refinement, which GCI for collection efficiency and pressure drop were, respectively, 3.08%, still a low and good value, and 17.40%, still a high value, but the lower obtained for the variable.

Now, comparing results between the two types of meshes with 287 thousand elements and expectations from empirical correlations, results were coherent to predictions of CFD. Collection efficiency is affected by wall effects, such as drag and friction, so it was expected a more reliable result of collection efficiency at the wall-refined mesh, as shown. As a variable equally distributed in the domain, a better accuracy for pressure is expected in meshes with regularly sized elements, as observed on results of pressure drop.

CONCLUSION

After analyzing the obtained results of the 12 simulated meshes it is conclusive that the proposed model correctly represents the expected flow profiles. Due to the critical region of the cyclone separator being located at walls and at vortex region, wall-refined meshes developed the expected trend needing fewer elements than regular element meshes. This caused less simulation time and machine effort, which was one of the objectives of this paper.

Even though the chosen mesh had a higher discretization error for collection efficiency, it was not a significant issue. Gains on accuracy obtained in a more refined mesh did not compensate the necessary time to simulate the device.

Nomenclature

- \( a \) Height of the cyclone inlet
- \( b \) Width of the cyclone inlet
- \( B \) Diameter of the particle matter outlet
- \( d_i \) Diameter of the cyclone central axis
- \( D_c \) Diameter of the cyclone body
- \( D_i \) Diameter of air outlet
- \( D_i \) Particle diameter
- \( \epsilon_a \) Approximated relative error
- \( \epsilon_{\text{ext}} \) Extrapolated relative error
- \( DPM \) Discrete phase model
- \( G \) Geometric parameter in Eq. (26)
- \( GCI \) Grid convergence index
- \( h \) Representative mesh size
- \( h_c \) Height of the cylindrical part of the cyclone
- \( H \) Cyclone total height
- \( K_x \) Dimensionless parameter in Eq. (27)
- \( K_y \) Dimensionless parameter in Eq. (28)
- \( K_z \) Dimensionless parameter in Eq. (29)
- \( LES \) Large Eddy simulation
- \( N \) Vortex exponent
- \( n_i \) Number of injected particles in the inlet
- \( n_{\text{trap}} \) Number of trapped particles in particle outlet
- \( N \) Total number of elements in the mesh
- \( \phi \) Apparent order of the \( P \) analyzed parameter
- \( P_i \) Pressure in the inlet of the cyclone
- \( P_{\text{out}} \) Pressure in the air outlet of the cyclone
- \( Q \) Gas volumetric flow rate
Grid refinement ratio
RSM Reynolds stress model
s Signal function
S Internal height of the air outlet
SIMPLE Semi-implicit method for pressure linked equations
T Temperature in K
u Fluctuation velocity
vi Gas velocity at the cyclone entry
Vnl Annular volume between S and Zc
Zc Vortex natural length

Greek
δ Relative error
ε Aggregated error
Δ Dimensionless parameter
ΔPcorrelation Pressure drop by empirical correlation
ΔP_{CFD} Pressure drop by CFD correlation
ΔV Volume of the i\textsuperscript{th} element of the mesh
η\textsubscript{correlation} Collection efficiency by empirical correlation
η\textsubscript{CFD} Collection efficiency by CFD correlation
µ Gas viscosity
ρ Gas density
ρp Particle density
σ Standard deviation
τ Relaxation time
φ Discrete solution
φ\textsuperscript{ext} Extrapolated discrete solution

Subscript
i Cartesian coordinate
j Cartesian coordinate
k Cartesian coordinate
in Inlet
t Tangential
z Z\textsuperscript{axis}
21 refers to GCI between meshes 2 and 1
32 refers to GCI between meshes 3 and 2
43 refers to GCI between meshes 4 and 3
54 refers to GCI between meshes 5 and 4
65 refers to GCI between meshes 6 and 5

REFERENCES

R.A.F. OLIVEIRA¹
G.H. JUSTI²
G.C. LOPES³

¹Graduate Program in Chemical Engineering, Federal University of São Carlos, Rodovia Washington Luís, São Carlos - São Paulo - Brazil
²Institute of Engineering, Federal University of Mato Grosso, Av. Fernando Corrêa da Costa Cuiabá - Mato Grosso - Brazil

NAUČNI RAD

PROUČAVANJE SEPARATORA CIKLONA KORIŠĆENJEM KONVERGENCIJE KOORDINATNE MREŽE RAZLIČITIH STRUKTURA

U projektovanju ciklona, dva značajna parametra za ocenu njegove praktične primenljivosti su smanjenje pritiska i efikasnost separacije. Optimalni dizajn pruža veću efikasnost i manji pad pritiska. U ovom radu, primenjene su nezavisne mreže kako bi se odredila najprikladnija mreža za simulaciju dvoфазног тока у Stairmand ciklonu. Alati прораčunske dinamike fluide (CFD) sa Eulerian-Lagrangian pristupom korišćeni su za simulaciju protoka. Иstražene су две различите структуре мређа, са неколико величина окала: једна са утицајем зида и друга са регуларним елементима. Metoda indeksа konvergencije mређе (GCI) je примена за процену независности реzultата. Резултати CFD модели упореđeni су са empirijskim korelациjамa iz literature и utvrđeno je добар спајање. Прва mређa са 287 hiljada elemenata дала je грешку од 9,8% за efikasnost separacije i 14,2% за pad pritiska, dok je ista mређa, сa регуларним elementима, dala grešku од 8,7% za efikasnost separacije i 0,01% за pad pritiska.

Кључне речи: CFD, ciklon, čađ otpadaka prerade šečerne repe, numerička mређa, indeks konvergencije mређе.

Ključne reči: CFD, ciklon, čađ otpadaka prerade šečerne repe, numerička mређa, indeks konvergencije mређе.