CFD ANALYSIS FOR DESIGN OPTIMIZATION OF
REVERSE FLOW TYPE CYCLONE SEPARATOR

Mr. SANTOSH T, Prof. R SURESH, Prof. K.V.SREENIVAS RAO and
Prof. U S MALLIKRJUN
1M.Tech. Student, 2Professors, Department of Mechanical Engineering,
Siddaganga Institute of Technology, Tumkur- 572 103

ABSTRACT

This study was concerned with the most common reverse flow type of cyclones where the flow enters the cyclone through a tangential inlet and leaves via an axial outlet pipe at the top of the cyclone. A cyclone separator for separating a solid particulate from the gas medium is described. The separator is provided with housing, an outlet for discharging the solid particulate separated from the medium, a pipe for evacuating the clean fluid from the housing and a swirling means capable of imparting vertical motion to the medium. The flow was assumed as unsteady, incompressible & isothermal and Numerical computations of the cyclones were studied. Due to the nature of cyclone flows, which exhibit highly curved streamlines and anisotropic turbulence, advanced turbulence models such as RSM (Reynolds Stress Model) have been used. The RSM simulation was performed using the commercial package FLUENT6.3.26

These calculations of the continuous phase flow were the basis for modeling the behavior of the solid particles in the cyclone separator. Velocities, pressures and the pressure drops have been studied in the present thesis. In the present work the pressures and velocities distribution have been generated using CFD. The pressure drops have been evaluated for the existing design and the modified design. Significant pressure drops have been observed in the optimized model.
1. INTRODUCTION

The cyclone separator is one of the most elegant pieces of engineering equipment. It is a device with no moving parts and virtually no maintenance. It enables particles of micrometers in size to be separated from a gas moving at about 15 m/s without excessive pressure-drop. Cyclone separators are very widely used throughout industry. Moreover, they can be found in all sizes and shapes. They can be used in some industries such as: Oil and gas, Power generation, Iron and steel industry, Cement plants, Coking plants, Coal fired boilers

A full understanding of how the cyclone separator works and how individual particles behave within it is not yet available. Little information has been gathered until the invention of the measuring equipment necessary to measure fluid velocities within cyclones (laser Doppler anemometry - LDA). Ultimately, the development of computational fluid dynamics (CFD) codes could accurately model swirling flows within the cyclone.

A cyclone separator is a device, which causes centrifugal separation of materials in a fluid flow. Unlike the slow settling of particles within a settling tank, a cyclone separator system yields fast separation and utilizes less space. Separation occurs quickly because one “g” of the gravitational force is replaced by multiple “g” of the acting centrifugal force, the material to be separated can consist of solid particles or liquids, i.e. droplets, which are classified according to size, shape, and density. The cyclone utilizes the energy obtained from the fluid pressure gradient to create rotational fluid motion. This rotational motion causes the dispersed phase to separate relatively fast due to the strong acting forces. In widely used reverse flow cyclones of the cylinder on cone design type, gases spiral down from a tangential inlet towards the apex of a conical section, where the flow is reversed and the particles are collected in a hopper. The continuous phase then proceeds upward in an inner core flow towards the gas
exit via the vortex finder. Cyclone designs have been developed over many years since their invention. Nowadays, there exist a large number of different types for various industrial applications. Many attempts have been made to improve the performance of cyclones by modifying their shape in terms of the ratio of different key dimensions. Normally, the continuous phase flow still carries some particles when it proceeds upward in the inner flow core towards the gas exit. Therefore, a solid apex cone has been incorporated in the cyclone to slow down the flow inside the dust collector (hopper).

2. PROBLEM DESCRIPTION

The gas flow in the cyclone separator is turbulent, and this creates a complication when using CFD. With direct numerical simulation (DNS) such CFD simulation were already carried out in small, simple geometries. This field is advancing fast as the computational power increases. However, with currently processing equipment, this is not possible yet. Therefore turbulence models are required. A recent turbulence model technique is Reynolds Stress Model (RSM). Therefore; RSM has been used for CFD Simulation.

In this thesis two cyclone separators with different inlet angles are studied. Predictions of the flow pattern, velocity and pressure drop in the cyclone separator are estimated by using CFD and particularly Reynolds Stress Model (RSM). For that, the specific goals of the work are the following:

- Calculation of the velocity profiles at various axial positions.
- Investigation of particle trajectories
- Calculation of the pressure drop for a given design

2.1 Particle Dynamics

Collection of solid or liquid particles in an air pollution control device is based upon the movement of a particle in the gas (fluid) stream. For a particle to be captured, the particle must be subjected to external forces large enough to
separate it from the gas stream. Forces acting on a particle include three major forces and other forces. They are

Gravitational force, Buoyant force, Drag force, other forces including magnetic, inertial, electrostatic, and thermal forces.

The consequence of acting forces on a particle results in the terminal velocity for a particle to settle. The terminal velocity (also known as the settling velocity) is a constant value of velocity reached when all forces (gravity, drag, buoyancy, etc.) acting on a body is balanced. The sum of all the forces is then equal to zero (no acceleration). To solve for an unknown particle settling velocity, the flow regime of particle motion must be determined. Once the flow regime has been determined, the settling velocity of a particle can be calculated.

2.2 Cyclone Separator

A cyclone is a conical vessel into which a dust-bearing gas-stream is passed tangentially. Because the rotating motion of the gas in the cyclone separator arises from its tangential entry and no additional energy is imparted within the separator body, a free vortex is established. The flow descends rotating near the wall, until a certain axial location where the axial velocity component reverses itself, thus making the flow to ascend. This is referred to as the vortex end position. The ascension proceeds near the cyclone axis and, since the flow rotation continues, a double vortex structure is formed, as indicated in the figure 1. The inner vortex finally leads the flow to exit through a central duct, called the vortex finder. The solids are thrown to the outside edge of the vessel by centrifugal action, and leave through a valve in the vortex of the cone.

Cyclones were originally used to clean up the dust-laden gases leaving simple dry process kilns. If, instead, the entire feed of raw mix is encouraged to pass through the cyclone, it is found that a very efficient heat exchange takes place: the gas is efficiently cooled, hence producing less waste of heat to the
atmosphere, and the raw mix is efficiently heated. This efficiency is further increased if a number of cyclones are connected in series.

2.3 Design Considerations

A small inlet and outlet therefore result in the separation of smaller particles. The depth and diameter of the body should be as large as possible because the former determines the radial component of the gas velocity and later controls the tangential component at any radius. In general the larger the particles, the larger should be the separator diameter, because greater is the radius at which they rotate, the greater too is the inlet velocity which can be used without causing turbulence within the separator. The factor which ultimately settles the maximum size is the cost. Because the separating power is directly related to the throughput of gas, the cyclone separator is not very flexible though its efficiency can be improved at low throughputs by restricting the area of the inlet with a damper and thereby increasing the velocity. However it is better to use a number of cyclones in parallel and to keep the load on each approximately the same.

Because the vertical component of the velocity in the cyclone is downwards everywhere outside the central core, the particles will rotate at a constant distance from the centre and move continuously downwards until they settle on the conical base of the plant. Continuous removal of the solids is desirable so that the particles do not get entrained again in the gas stream due to the relatively low pressures in the central core. Entrainment is reduced to a minimum if the separator has a deep conical base of small angle. The effect of the arrangement and size of the gas inlet and outlet has been investigated and it has been found that the inlet angle should be of the order of 180°, as indicated in the figure 2. Further the depth of the inlet pipe should be small and a square section is generally preferable to a circular one because a greater area is then obtained for
a given depth. The outlet pipe should extend downwards well below the inlet in order to prevent short-circuiting.

The efficiency of the cyclone separator is greater for large than for small particles and it increases with the throughput until the point is reached where excessive turbulence is created. Figure 3 shows the efficiency of collection plotted against particle size for an experiment separator for which the theoretical “cut” occurs at about 10µm. It may be noted that an appreciable quantity of fine material is collected, largely as a result of agglomeration, and that some of the coarse material is lost with the result that a sharp cut is not obtained.

2.4 Cyclone Performance

From an engineering point of view, cyclone performance is measured by collection efficiency (The fraction of solids separated) and pressure drop.
**Pressure drop:** The pressure drop across a cyclone is an important parameter to the purchaser of such equipment. Increased pressure drop means greater costs for power to move exhaust gas through the control device. With cyclones, an increase in pressure drop usually means that there will be an improvement in collection efficiency (one exception to this is the use of pressure recovery devices attached to the exit tube; these reduce the pressure drop but do not adversely affect collection efficiency). For these reasons, there have been many attempts to predict pressure drops from design variables. The idea is that having such an equation, one could work backwards and optimize the design of new cyclones.

The pressure drop across a cyclone consists of a combination of local inertia-related losses and a frictional loss. The local losses include an expansion loss at the cyclone inlet and a contraction loss at the entrance of the vortex finder. The frictional loss includes a swirling loss due to the friction between the gas flow and the cyclone wall, and a friction loss of the gas flow in the outlet. In most cases, the contraction loss at the entrance of the vortex finder and the friction loss associated with the swirling motion of vortices are the major factors.

**Collection Efficiency:** A number of formulations have been developed for determining the fractional cyclone efficiency for a given size particle. Fractional efficiency is defined as the fraction of particles of a given size collected in the cyclone, compared to those of that size going into the cyclone.

Collection efficiency = \( \frac{(\text{inlet loading} - \text{outlet loading})}{(\text{inlet loading})} \times 100 \)
3. RESULTS AND DISCUSSIONS

3.1. Steps followed during the execution of project:

**Phase 1: (Initial Model)**

3D Model generation as in figure 4 & 5. Mesh generation, Solution, Post Processing

**Phase 2: (Modified Model)**

3D Model generation based on CFD results of Initial Model as in figure 8 & 9. Mesh generation, Solution, Post Processing

3.2. Geometric Details

<table>
<thead>
<tr>
<th>Particulars</th>
<th>Dimensions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Width (m)</td>
<td>0.77</td>
</tr>
<tr>
<td>Inlet Height (m)</td>
<td>1.50</td>
</tr>
<tr>
<td>Exit Pipe Length (m)</td>
<td>1.81</td>
</tr>
<tr>
<td>Material Outlet Dia (m)</td>
<td>0.25</td>
</tr>
<tr>
<td>Gas Outlet Dia (m)</td>
<td>1.32</td>
</tr>
<tr>
<td>Cone Length (m)</td>
<td>2.09</td>
</tr>
<tr>
<td>Cylinder Length (m)</td>
<td>2.37</td>
</tr>
</tbody>
</table>

Table 1: Geometric Details of Cyclone Separator

Fig 4: Geometric Details of Cyclone Separator Half model is considered for analysis due to symmetry

Fig 5: Symmetric Model considered for analysis
3.3 Results of Initial Model

3.3.1 Pressure Contours of mixture at mid plane & top plane of cyclone (Pa)

From the figure 6 (left) at mid plane shows the variation of Pressure at mid plane and following readings are drawn:

Pressure at the inlet in the initial model observed is -2670 Pa.

Pressure at the outlet in the initial model observed is -5020 Pa.

Pressure drop in the initial model from inlet to outlet is =-5020-(-2670)= -2350 Pa

From the figure 6 (right) at top plane shows the variation of Pressure at different heights of the cyclone geometry.

Pressure at the inlet in the initial model observed is -2670 Pa

Pressure at the vertex finder in the initial model observed is -5810 Pa

Pressure drop from the inlet to vertex finder in initial model is -5810-(-2670) = -3140 Pa
3.3.2 Velocity Contours of mixture at mid plane of cyclone assembly (m/s)

From the figure 7 (left) at mid plane shows the variation of Velocities at mid plane and the following readings are drawn.

Velocity at the vertex finder in the initial model observed is 48.7 m/s

From the figure 7 (right) at top plane shows the variation of velocities at different heights of the cyclone geometry.

Velocity at the inlet in the initial model observed is 19.5 m/s

3.4 Results of Modified Model

3.4.1 Geometry Details of Modified Model

![Fig 8: Geometric Details of Cyclone Separator-Modified Model Half model is considered for analysis due to symmetry](image1)

![Fig 9: Symmetric Half Modified Model considered for analysis](image2)

3.4.2 Pressure Contours of mixture at mid plane & top plane of cyclone (Pa)

From the figure 10 (left) at mid plane shows the variation of Pressure at mid plane and following readings are drawn
Pressure at the Inlet of the cyclone in the modified model is -2670 Pa.

Pressure at the Outlet of the cyclone in the modified model is -4460 Pa.

Pressure drop in the modified model from inlet to Outlet is -4460 - (-2670) = -1790 Pa

From the figure 10 (right) at top plane shows the variation of Pressure at different heights of the cyclone geometry.

Pressure at the inlet in the modified model observed is -2670 Pa

Pressure at the vertex finder in the modified model observed is -5230 Pa

Pressure drop from inlet to vertex finder in modified model is 5230 - (-2670) = -2560 Pa

3.4.3 Velocity Contours of mixture at mid plane of cyclone assembly (m/s)

From the figure 11 (left) at mid plane shows the variation of Velocities at mid plane and the following readings are drawn.

Velocity at the vertex finder in the modified model observed is 43.7 m/s

From the figure 11 (right) at top plane shows the variation of velocities at different heights of the cyclone geometry.

Velocity at the inlet in the modified model observed is 15.8 m/s

![Fig 10: Pressure Contours of mixture at mid plane & top plane of cyclone for Modified Model (Pa)](image1)

![Fig 11: Velocity Contours of mixture at mid plane & top plane of cyclone (m/s)](image2)
3.5 Comparison of Results from Initial & Modified Models

Pressure Contours of mixture at mid plane & top plane of cyclone (Pa) in Initial & Modified Model

- Pressure drop in the initial model observed is 2350 Pa. Pressure drop in the Modified model observed is 1790 Pa. Due to design change the Pressure drop is reduced by 560 Pa in the Modified model. Hence, the modified design can be implemented for the industrial applications.

- Pressure drop from the inlet to vertex finder in the initial model observed is 3140 Pa. Pressure drop from the inlet to vertex finder in the Modified model observed is 2560 Pa. Due to design change the Pressure drop is reduced by 580 Pa

Velocity Contours of mixture at mid plane of cyclone assembly (m/s) in Initial & Modified Model

- Velocity at the inlet in the initial model observed is 19.5 m/s, Velocity at the inlet in the modified model observed is 15.8 m/s.Due to change in the inlet angle and due to increase in the inlet surface area.

- Velocity at the vertex finder in the initial model observed is 48.7 m/s. Velocity at the vertex finder in the modified model observed is 43.7 m/s. Velocity is reduced in the modified model by 5 m/s due to change in the inlet angle and due to change in the inlet surface area.

4. CONCLUSION

The nature of the gas flow of a particle in cyclone separator is highly swirling with anisotropic turbulence. Therefore, advanced turbulence models such as RSM and LES have to be applied to predict the gas flow behavior rather than the meanwhile classical $k – \varepsilon$ turbulence model.
The increase of gas inlet velocity will increase the separation efficiency, but it will also increase the pressure drop. In order to get the more efficiency of cyclone separator the pressure drop has to be reduced across intake and exit.

To achieve the lesser pressure drop, initial model inlet is modified and maintained some angularity through which cement particles and gaseous mixtures are entering into the cyclone separator.

Velocity at the entry of the vertex finder in the initial and modified model observed is 48.7 m/s and 43.7 m/s respectively. In the optimized model from the CFD results, it is identified that the velocity has been reduced by 5 m/s.

Pressure drop from inlet to outlet in the initial and modified model observed is 2350 Pa and 1790 Pa respectively. In the optimized model from the CFD results, it is identified that the Pressure drop is reduced by 560 Pa. Hence the modified model with little modification at the inlet angle can be proposed for the industrial applications.

The proposed model provides a convenient way to study the effects of variables related to operational conditions, cyclone geometry. The particle properties play an important role to optimize the design and control of cyclone process.

5. BIBLIOGRAPHY


6. HARWOOD, R. SLACK, M. CFD analysis of a cyclone, QNET-CFD Network Newsletter, Volume 1, No.4 – November 2002


