Simulation and Experimental Validation of an Axial-Flow Hydrocyclone

Oliver A. Oakman*

UNSW Canberra at ADFA.

Axial-flow hydrocyclone separators are a potentially viable alternative to the conventional reverse-flow designs; particularly on the smaller scale where a large pressure loss in detrimental. An experimentally validated Large-Eddy-Simulation model of a cylindrical axial-flow miniature hydrocyclone has been developed. Using this model, an investigation into the flow structure of the hydrocyclone has been conducted, and an investigation into separation efficiency and pressure characteristics has been conducted as part of the experimental validation. It has been found that these devices can potentially separate particles with a significantly lower pressure drop than the conventional reverse-flow hydrocyclones due to their high intensity swirl and simpler flow structure.

Contents

I. Introduction

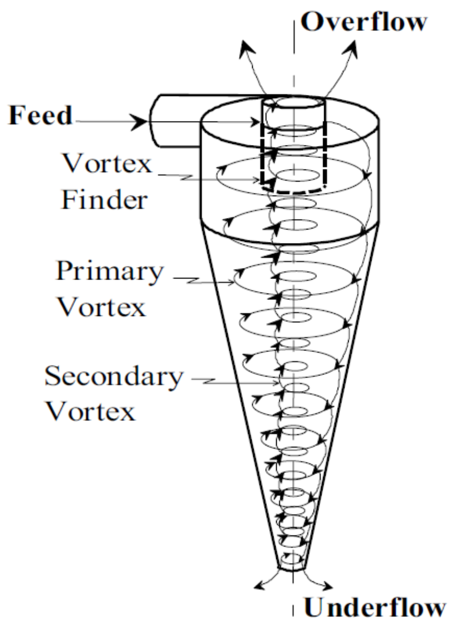
It is often desirable to separate a fluid mixture into its components for industrial applications. Cyclonic separators are devices that separate fluid into its constituent parts; for example separating water droplets from oil, or dust particles from air. These are respectively defined as the dispersed phase and the continuous phase. They are primarily used as pre-filters for gas intake systems to decrease maintenance costs and increase life, and also to capture valuable minerals from a mining slurry.

The cyclonic separator is beneficial compared to centrifuges or gravity-based separation due to their ability to operate continuously. The induction of high-intensity swirl enacts centrifugal force on the dispersed phase, moving them to the outer swirl, given the drag force on a particle is proportional to the square of

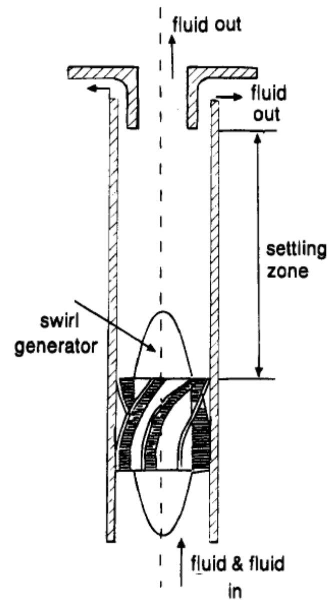
*School of Engineering and Information Technology, ZEIT4500/4501

diameter, though centrifugal is proportional to D^3. There is also higher reliability in the use of these devices as the separation mechanism is based on geometry and fluid mechanics, rather than mechanical interaction.

An alternative design to the industry standard reverse-flow design is the axial-flow hydrocyclone. Cyclonic dust separators in pre-filters have commonly been of this design (Stenhouse and Trow, 1979). This design is characterised by concentric outlets and lacks a reversed helical flow in the core of the device. While the outlet design has been simplified in this representation, the crux of axial-flow hydrocyclone design is the management of the concentric outlets. These hydrocyclones have been shown to have viable application to gas-liquid and liquid-liquid separation (Dickinson, 1998; Gauthier et al., 1992; Swanborn, 1988), however they have not seen broad use in industry despite their advantages. There is less chance of re-entrainment in the outlets as there are no reversed interfacing flow in the body, and they also have a theoretically lower pressure drop without sacrificing separation efficiency due to a much less disturbed flow field in comparison to reverse-flow (Nieuwstadt, 1995). Examples of these two designs are shown in figure 1.



(a) A conventional reverse-flow hydrocyclone with a tangential inlet (Medronho, 2003)



(b) An axial-flow hydrocyclone with a swirling vane (Nieuwstadt, 1995)

Figure 1: A example of a reverse-flow hydrocyclone with a conical section and a cylindrical axial-flow hydrocyclone

I.A. Project Statement

While reverse-flow mini-hydrocyclones have found application in fine particle separation, the large pressure drop in these devices poses difficulties in maintaining a reasonable flow rate; particularly if it is within a self contained chip. The replacement of these hydrocyclones with the axial-flow design could reduce the required pump capacity. Viability at this level could also have a knock-on effect in the large scale industry by using the axial-flow design and reducing the pressure loss through hydrocyclones.

This project is a proof-of-concept investigation into the viability of using axial-flow mini-hydrocyclones for fine particle separation as opposed to the reverse-flow design. If the hydrocyclone design investigated were to achieve a comparable separation efficiency at a lower pressure drop as hypothesised, it could improve mini- and micro- device efficiency. The overall aim of this thesis is to analyse and develop an understanding of the flow field in an axial flow hydrocyclone using an experimentally validated numerical model. Of particular interest are the pressure drop and separation efficiency for various flow rates, which will be investigated experimentally.

II. Numerical Model

To analyse the highly complex flow characteristics of the axial-flow hydrocyclone, it is far more practical to utilise computer simulation rather than experimental methods (such as injecting dye and observing the streamlines). As the hydrocyclone design in question is small, this adds extra difficulty in experimental analysis. Thus, computational fluid dynamics (CFD) was used in order to analyse the flow field.

II.A. Design and Meshing

As the axial-flow hydrocyclone has not been thoroughly researched, it was necessary to simplify the design as much as possible. It was also necessary to design an axial-flow hydrocyclone that could be reasonably compared to known results from a previous study (Zahra, 2012). A cylindrical hydrocyclone with a tangential inlet and tangential outlet for the underflow was designed in order to induce high-intensity swirl and also for construction simplicity.

Due to the simplistic geometry of the hydrocyclone, a structured mesh was created, dominated by hex cells. A structured mesh tends to converge faster, particularly when the cells are oriented along the expected flow path. The skewed cells at the interface between the tangential inlet and outlet did not cause any numerical instability. Figure 2 shows this mesh.

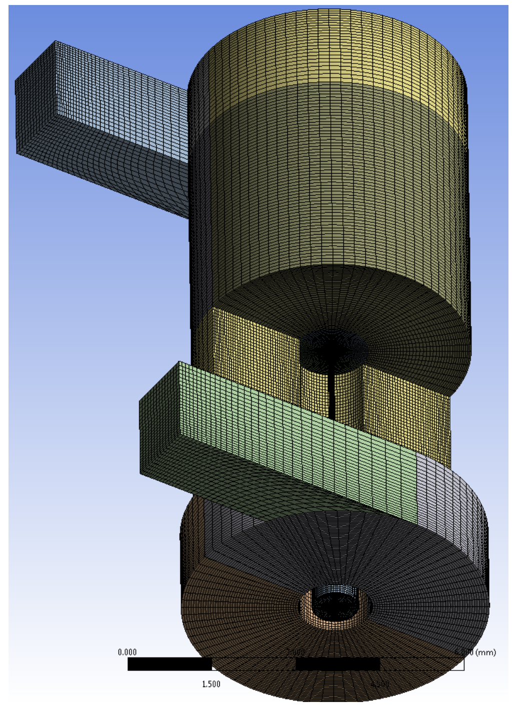


Figure 2: Mesh from the base, showing the vortex finder outlet and the internal elements of the mesh. Note the inner-most elements are unstructured, though they make up a small percentage of the mesh

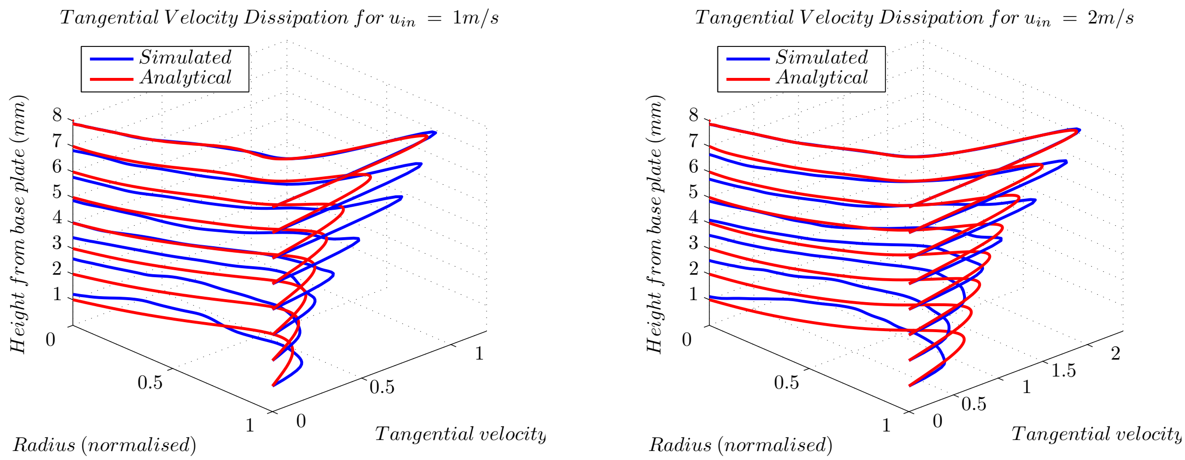
II.B. Simulation Method

The potential for Large-Eddy Simulation (LES) in the numerical modelling of cyclone separators was investigated by Shalaby (2007) and it was found that LES captured the dynamic behaviour of the cyclone, and showed a satisfying agreement with experimental results when Eulerian-Eulerian one-way particle coupling was imposed. LES was also demonstrated to be in agreement with Direct Numerical Simulation (DNS) models, the most computationally expensive and theoretically most accurate model. The LES model is the most sensible choice for modelling hydrocyclones when large computing power is available. This model directly solves the largest eddies in the flow field and models the sub-grid eddies, which are time-dependent. Thus, a transient model was required. For this study, ANSYS Fluent 14.5 commercial code was used.

II.C. Verification

A common method for determining if the mesh is adequate for the physics of the situation is by conducting a grid independence study. For this mesh, however, this was not feasible as subdividing the mesh another level interfered with the boundary layer approximation, resulting in a non-physical acceleration in the outlet. In order to determine if the simulation was physically reasonable, an analytical study by Talbot (1954) was utilised. This study developed an expression for the dissipation of tangential velocity in laminar swirling pipe flow. Figure 3 shows the comparison between the simulated results for two inlet velocities, and the respective analytical prediction. These compare quite reasonably given the assumptions made and the simulation method was considered to be verified. This method is outlined below, where $W(r,z)$ is the tangential flow profile along the radius at a pipe height z;

$$W(r,z) = \sum_{n=1}^{\infty} C_n J_1(\lambda_n r) e^{\frac{\lambda_n{}^2}{\mathrm{Re}_z} z} \tag{1}$$



(a) Verification of $V_{in} = 1m/s$

(b) Verification of $V_{in} = 2m/s$

Figure 3: Verification using analytical predication of tangential velocity dissipation

where $J_1(\lambda_n r)$ is the Bessel function of the first kind, with eigenvalues (which are well known), Re_z is the Reynolds number in the axial direction, a is the radius in metres $0 \leq r \leq \frac{r}{a}$ and $0 \leq z \leq \frac{z}{a}$, and;

$$C_n = \frac{\int_0^1 rW(r,0)J_1(\lambda_n r)dr}{\int_0^1 rJ_1^2(\lambda_n r)dr} \tag{2}$$

Similar to the spacial discretisation, the temporal discretisation needed to be verified. First, the stability was analysed. To do this, the Courant-Friedrichs-Lewy condition was used to determine a suitable time-step for the numerical model. Best practice in (CFD) is to ensure the Courant number is less than or equal to one. That is, for the maximum case (in the dimension of the maximum velocity, being $-x$);

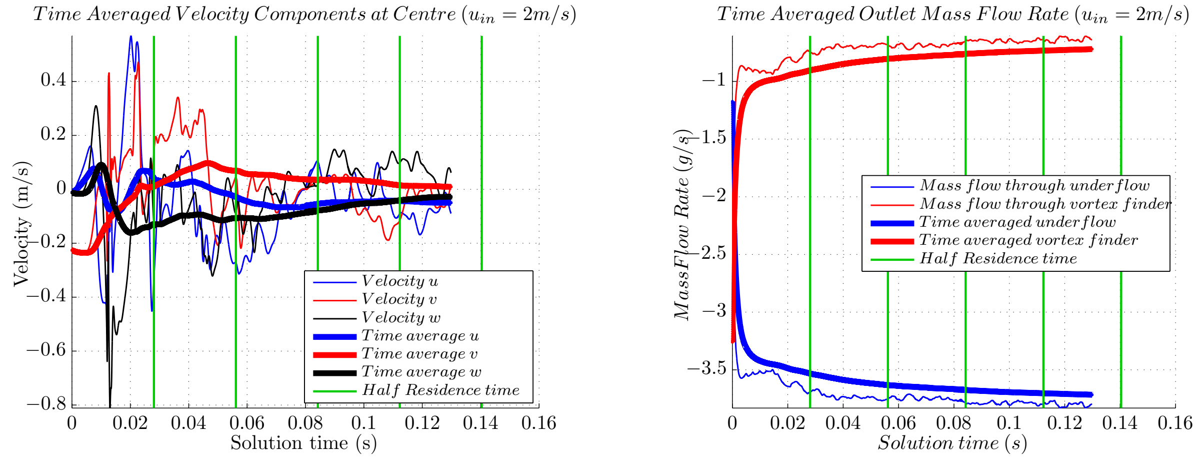
$$C_{\max} = \frac{u_{\max}\Delta t_{\max}}{\Delta x_{\max}} \leq 1 \tag{3}$$

where u_{max} is the maximum velocity, $\Delta t_{\max}$ is the maximum allowable time-step, and $\Delta x_{\max}$ is the largest grid spacing. By analysing the mesh, an approximate yet conservative estimate of $\Delta x_{\max} = 15 \times 10^{-6}$m was determined. Setting the Courant number to 0.9, the maximum time-step for each case was calculated as;

$$\Delta t_{\max} = \frac{13.5 \times 10^{-6}}{u_{\max}} \text{where} C \leq 1 \tag{4}$$

which ensured that the transient simulation would be stable.

Second, the transient solution had to run for long enough to represent a statistically significant time-average. As particles were being tracked with the transient solution, it was also necessary to ensure an appropriate amount of particle residence times were run. This is the average time for an injected particle to reach the outlet, $t_{res} = \frac{V}{Q}$. Approximately 2.3 residence times were simulated to ensure particles were ejected and the time-average of the solution was meaningful. Figure 4a shows the time-averaging of the velocity components at the centre of the hydrocyclone, and figure 4b shows the mass flow rate from the underflow and vortex finder outlet. These both converge to statistically small variations with increasing time.



(a) Measured and time-averaged velocity components

(b) Measured and time-averaged mass flow rate

Figure 4: Temporal verification using velocity and mass flow rate monitors

III. Experimental Validation

It is necessary in CFD to validate numerical models against experimental results or expected behaviours. In this project, an experiment was developed to compare the pressure drop and volumetric split of the hydrocyclone to the simulated data.

III.A. Experimental Method

The hydrocyclone was constructed using micro-milled perspex plates which were bolted together. It was manufactured with respect to the designed model and is a good representation of it. The major macroscopic difference was the body height, with an error of 5%. The plates were observed under an optical light microscope and it was found that it was constructed accurately, however the vortex finder outlet was 13% larger than the designed model. This variation did not appear to have any adverse affect on the results.

Using three different solid soda lime glass micro-sphere particle sources; Prizmalite P2015SL, Potters AI and Potters AH, a sample was created which covered the largest possible range of diameters. This sample (by mass) was; $1.2 \times m_{(\text{Cospheric2015})} + 1.0 \times m_{(\text{PottersAH})} + 1.0 \times m_{(\text{PottersAI})}$. Preparing 0.2% volume (0.502% mass) samples, particles were processed through the hydrocyclone into beakers. The water was then boiled off and the particles were collected for size and mass analysis. During these runs, volumetric split ratio was noted, and pressure at the inlet was measured.

III.B. Experimental Uncertainty Analysis

Due to the scale of the hydrocyclone, slight variations in flow and pressure were significant. To mitigate this, multiple inlet pressure readings were taken, and several intermediate times and volumes were recorded to ascertain the flow rate. An investigation into pressure hysteresis was also conducted. There were subtle differences between raising and lowering the flow rate, however the degree of this hysteresis was small and did not create significant error.

It was also anticipated that the particle analysis would be highly scrutinised as it is very sensitive to optical properties; being refractive indexes and absorptivity. Lacking a spectrometer, the absorptivity of the glass particles was determined by inputting a known concentration by volume through the Malvern Mastersizer 2000 Particle Size Analyser and tweaking the absorptivity value until the concentration was in agreement. It was found that the absorptivity of the particles were 0.007 ± 0.0005. Using this value, particle distributions of the Potters AH were analysed which also shows good agreement with an optical measurement of the particles using a scanning electron microscope.

IV. Results and Validation

IV.A. Experimental Results

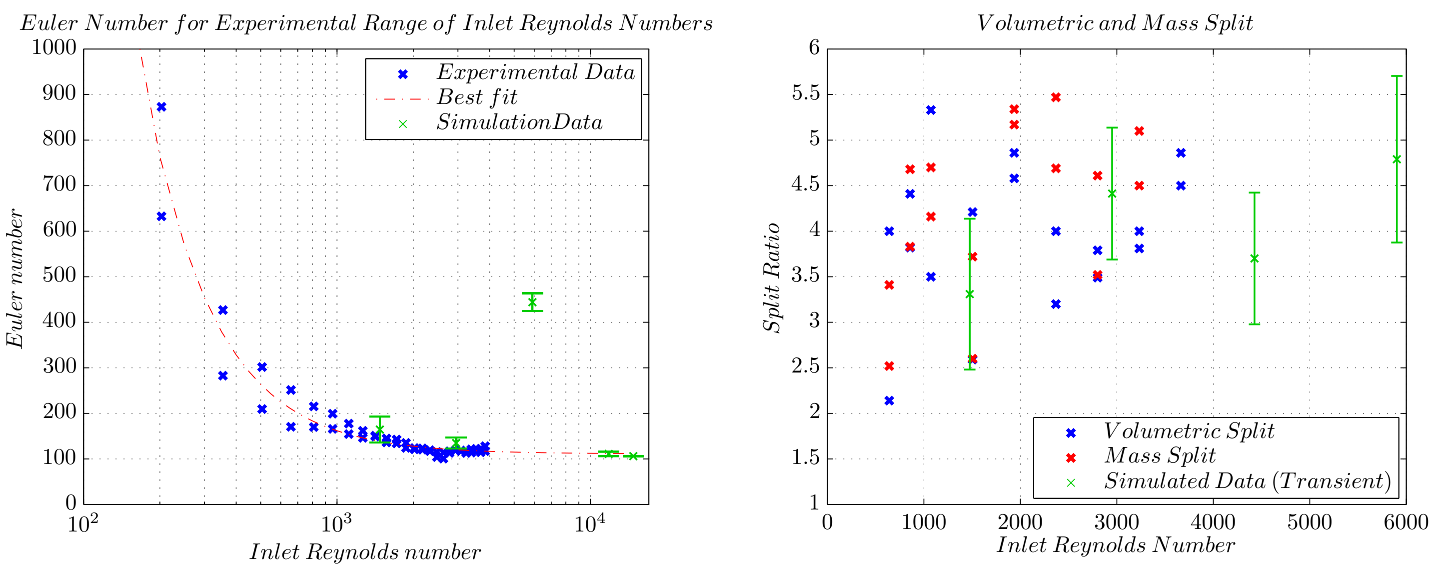
To validate the model, pressure data and volumetric split data were compared between experiment and simulation. Due to maintenance issues, the pump used was only able to reach an inlet velocity of $\sim 2.6m/s$. Figure 5 shows the experimental results superimposed with the simulated results with errorbars to indicate transient variation within two standard deviations. This shows a good agreement between experiment and the numerical model.

The Euler number is known as the pressure coefficient, and is a dimensionless measure of the pressure loss related to the kinetic energy input of the control volume. It is defined as;

$$Eu = \frac{2\Delta P}{\rho_c v_{ch}^2} \text{ where } v_{ch} = \frac{4Q}{\pi D^2} \quad (5)$$

For reverse-flow hydrocyclones, the Euler number is expected to approach a constant value with increasing Reynolds number. This value is the Euler number of the hydrocyclone design and for the reverse-flow designs, it tends to be greater than 1000 (Vieira et al., 2005).

The Euler number for simulation has an outlier at $Re \approx 4400$, equating to $u_{in} = 4m/s$, which is not valid. Otherwise, the simulation data was in good agreement with the experiment. The results show a low Euler number (~ 110) in comparison to standard reverse-flow hydrocyclones which indicates that the axial-flow hydrocyclone can potentially separate particles at a very low pressure loss. It was found that at the maximum experimental flow rate, particles began to separate at a low efficiency. This likely means that the hydrocyclone will separate particles at a higher flow rate.

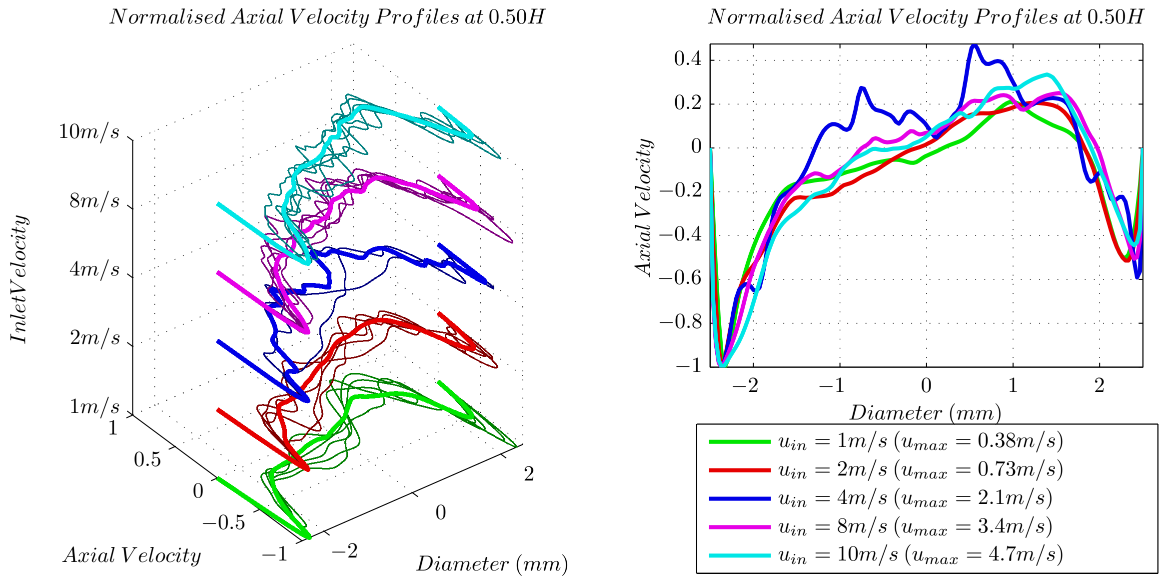


(a) Euler Number; the dimensionless pressure loss

(b) Volumetric split with errorbars on the simulation results to represent the transient data

Figure 5: Comparison of results between experiment and numerical model

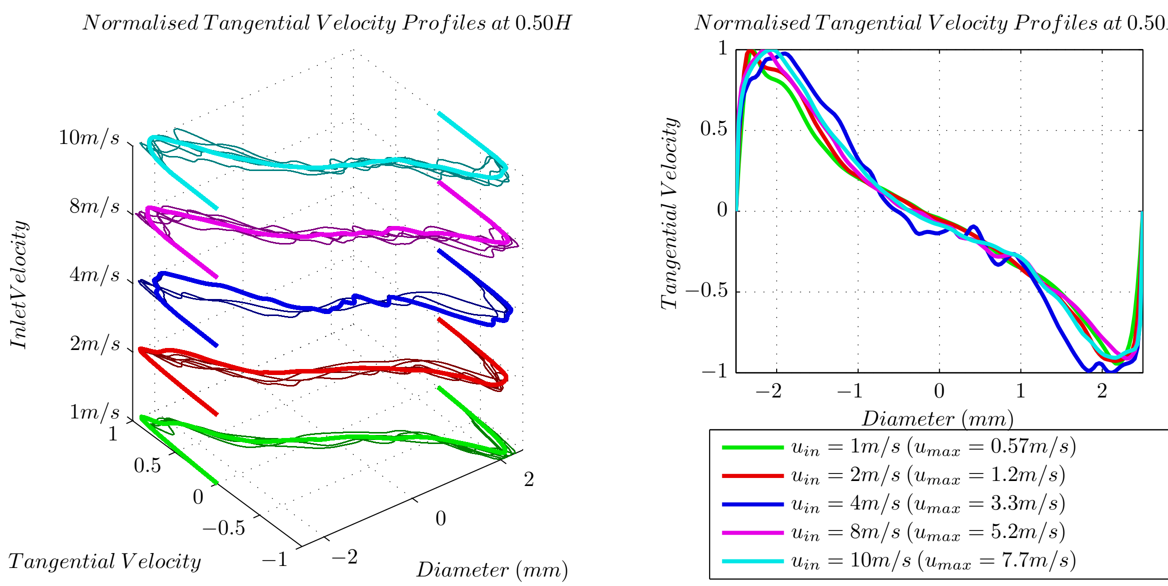
IV.B. Numerical Results



(a) Time dependent data (thin lines) and time-averaged data (thick lines)

(b) Comparison of normalised time-averaged tangential velocity profiles

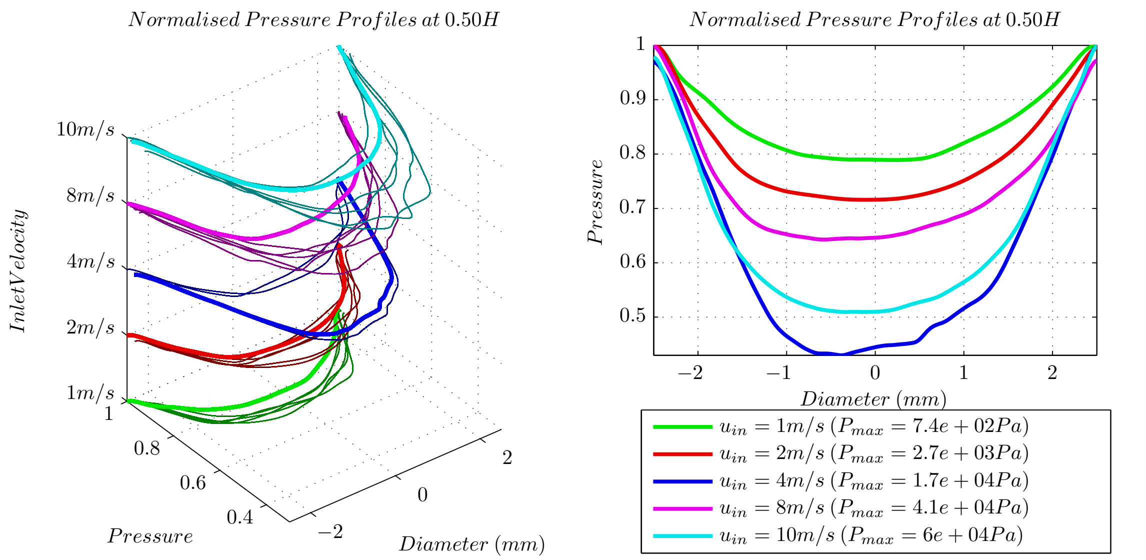
Figure 6: Axial velocity profiles at 50% cyclone body height



(a) Time dependent data (thin lines) and time-averaged data (thick lines)

(b) Comparison of normalised time-averaged tangential velocity profiles

Figure 7: Tangential velocity profiles at 50% cyclone body height



(a) Time dependent data (thin lines) and time-averaged data (thick lines)

(b) Comparison of normalised time-averaged pressure profiles

Figure 8: Pressure profiles at 50% cyclone body height

It can be seen that the normalised flow profiles are relatively similar. Axially, the hydrocyclone exhibits a reversed flow in the core (noting that negative axial velocity is travelling along the body). This characteristic is likely based on the pressure build-up near the hydrocyclone outlets that is seen in reverse-flow hydrocyclones, which then direct the inner helical flow up. In this hydrocyclone, it also exhibits some swirl about the x direction which potentially adds some particle re-entrainment in the flow. The velocity vector plots and the locus of zero axial velocity surfaces in figure 9 shows this.

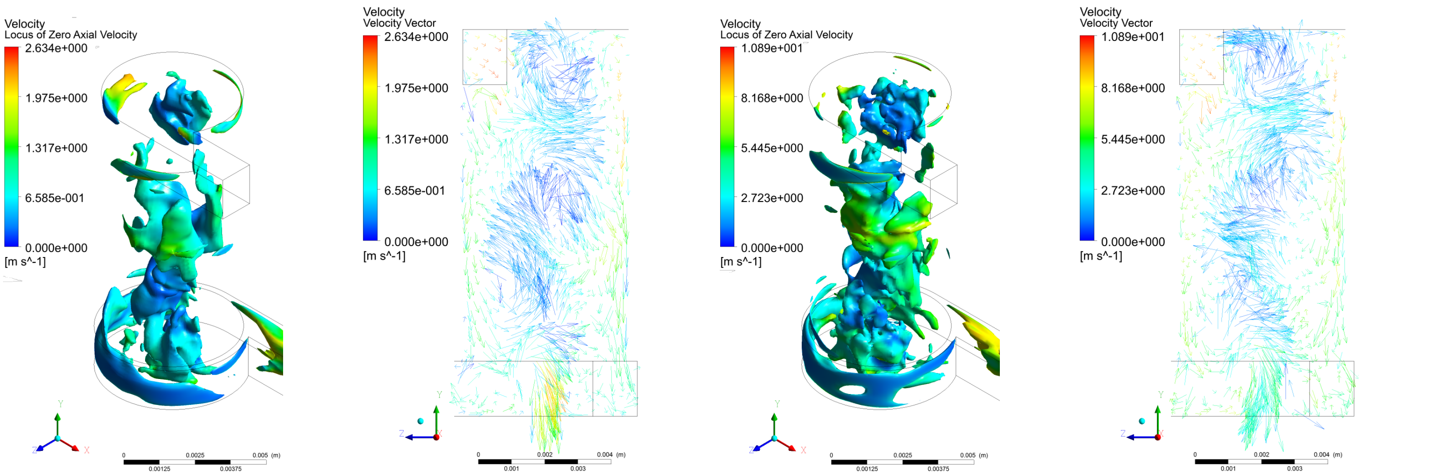
Radially, the hydrocyclone is significantly unstable with time and shows no inlet velocity dependent trends. These profiles have not been included due to this. It is likely that this velocity component oscillates and is subject to small instabilities, given that the radial component is itself quite small ($< 5\%$ of the inlet). The axial velocity profiles of the axial-flow hydrocyclone are very similar to the cylindrical reverse-flow hydrocyclone investigated by Zahra (2012). The tangential velocity, however, has a significantly larger peak velocity on the outside of the cyclone, creating higher swirl intensity and increasing centrifugal force on the particles. This mechanic is likely primarily what causes particle separation in these devices given the simpler flow structure.

Of particular interest is the pressure gradient which is a major component in separation of particles. As flow rate increases, the pressure gradient becomes larger and influences more of the flow in the core. The Euler number shows that the pressure gradient is proportional to the square of the characteristic velocity, such that $\frac{\partial P}{\partial r} \propto \rho_c v_{ch}^2$. This proportionality has reflected the pressure gradients in the hydrocyclone to within a reasonable degree given the approximation made (within 30%). It predicts the increase in pressure gradient with the trend expected, though as it does not consider the turbulence in the model, the approximation is inaccurate.

It can also be seen that the Reynolds number for transition for the axial-flow hydrocyclone is between 2000 and 3000. Reverse-flow hydrocyclones typically become turbulent when $Re > 500$. This is likely a major reason for the low Euler number for the device, as the flow field is more analogous to a standard pipe flow. The full pressure contours for $u_{in} = 1m/s$, $u_{in} = 2m/s$, $u_{in} = 8m/s$ and $u_{in} = 10m/s$ are shown in figure 10. As the vortex finder outlet is open to the atmosphere, there is a danger that the negative pressure in it could entrain air upwards and create an air core in the centre. In reverse-flow hydrocyclones, if the underflow and overflow outlets were both exposed to the atmosphere, this would occur. However, as there is no clear streamline from the vortex finder to either the underflow outlet or the inlet in the axial-flow

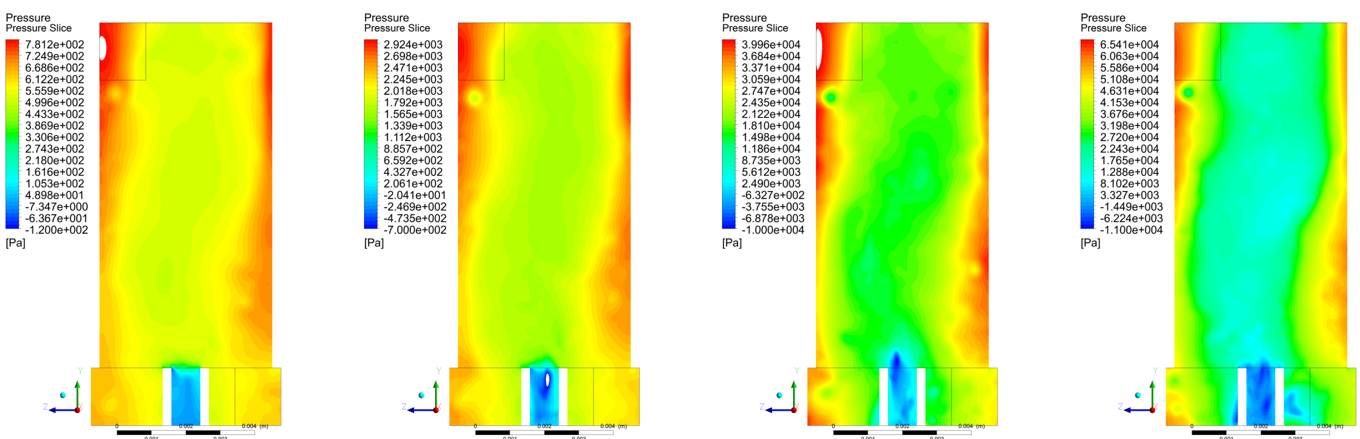
hydrocyclone, this was not likely to occur.

The source of the numerical error in the $u_{in} = 4m/s$ case is unknown, as the results appear to be verified. They have been included in figures 6 through 8 for completeness, however they do not exhibit reasonable profiles. It can be seen in figure 8a that the time dependent results do not show a good agreement with the time average, and as such the instability is likely due to the temporal discretisation.



(a) Locus of Zero Axial Velocity for $u_{in} = 2m/s$ (b) Velocity vectors (coloured by magnitude) for $u_{in} = 2m/s$ (c) Locus of Zero Axial Velocity for $u_{in} = 8m/s$ (d) Velocity vectors (coloured by magnitude) for $u_{in} = 8m/s$

Figure 9: Velocity Characteristics for the flow field



(a) Pressure contour for $u_{in} = 1m/s$ (b) Pressure contour for $u_{in} = 2m/s$ (c) Pressure contour for $u_{in} = 8m/s$ (d) Pressure contour for $u_{in} = 10m/s$

Figure 10: Pressure contours for the YZ plane of the hydrocyclone

V. Conclusions and Recommendations

An experimentally validated numerical model of a cyclindrical axial-flow hydrocyclone has been created. Through experimental analysis, pressure characteristics have been determined and it was found that they can operate at far lower pressure drops than reverse-flow hydrocyclones while potentially separating particles. This in shown through the small Euler number representing ‘cost of separation‘.

Similarly, velocity characteristics have been investigated and compared and contrasted to a similar reverse-flow hydrocyclone. The flow field appears to be more analogous to swirling pipe flow with high tangential velocity profiles at the outer diameter of the hydrocyclone. The hydrocyclone also appears to develop a low pressure core in the centre

It is recommended that a full separation analysis be completed by increasing the flow rate through the hydrocyclone. Further to this, design parameters should be investigated. It would be useful to investigate the addition of a conical section as is in most hydrocyclones. It is also recommended that a design analysis be conducted on the inlets and outlets of the axial-flow hydrocyclone, such as determining the effect of inlet to outlet area ratio on the pressure and volumetric split ratio, or the effect of inlet aspect ratio on the separation efficiency of the device.

Acknowledgements

The author would like to acknowledge Dr Jong-Leng Liow for his guidance and expertise on the subject, despite being on sabbatical for the initial few months of the project, Dr John Young for his guidance using the ANSYS Package software and the overall CFD course he taught, providing a foundation for numerical simulation theory, and Mr. Alan Fien for his much appreciated assistance in the experimental section of the project.

References

[1]Stenhouse, J. I. T. and Trow, M., "The Behaviour of Uniflow Cyclones," *The Second World Filtration Congress*, 1979.

[2]Dickinson, P., *Gas-Liquid Separation Within a Novel Axial Flow Cyclone Separator*, Ph.D. thesis, School of Mechanical Engineering, Cranfield University, Cranfield, Bedford, United Kingdom, 1998.

[3]Gauthier, T. A., Briens, C. L., and Bergougnou, M. A., "Uniflow Cyclone Efficiency Study," *Powder Technology*, Vol. 62, 1990, pp. 217–225.

[4]Swanborn, R. A., *A New Approach to the Design of Gas-Liquid Separators for the Oil Industry*, Ph.D. thesis, Civil Engineering and Geosciences Faculty, Delft University of Technology, Delft, CN, Netherlands, 1988.

[5]Nieuwstadt, F. and Dirkzwager, M., "A Fluid Mechanics Model for an Axial Cyclone Separator," *Industrial and Chemistry Research*, Vol. 34, No. 10, 1995, pp. 3399–3404.

[6]Medronho, R. A., Schuetze, J., and Deckwer, W.-D., "Numerical Simulation of Hydrocyclones for Cell Separation," *Latin America Applied Research*, Vol. 35, 2003, pp. 1–8.

[7]Zahra, J. P., *CFD Modelling of the Behaviour of a Cylindrical Miniature Hydrocyclone*, Undergraduate honors thesis, School of Engineering and Information Technology, UNSW Canberra at ADFA, Australia, 2012.

[8]Zhu, G., Liow, J.-L., and Neely, A., "Computational Study of Flow in a Micro-Sized Hydrocyclone," *17th Australiasian Fluid Mechanics Conference,*, Auckland, New Zealand, 2010.

[9]Shalaby, H. H., *On the Potential of Large Eddy Simulation to Simulate Cyclone Separators*, Ph.D. thesis, Department of Mechanics, Chemnitz University of Technology, Chemnitz, Germany, 2007.

[10]Talbot, L., "Laminar Swirling Pipe Flow," *Journal of Applied Mechanics*, Vol. 21, 1954, pp. 1–7.

[11]Vieira, L. G. M., Barbosa, E. A., Damasceno, J. J. R., and Barrozo, M. A. S., "Performance Analysis and Design of Filterning Hydrocyclones," *Brazilian Journal of Chemical Engineering*, Vol. 22, No. 1, 2005, pp. 143–152.

Appendix A: Flow Profiles

(See attachment)