DEVELOPMENT OF COMPUTATIONAL METHODS FOR INLET OPTIMISATION IN CENTRIFUGAL VENTILATION UNITS

Daniel Mitchell OWEN, Luke Andrew TEMPLE

Elta Fans Ltd, Engineering Department, 46 Third Avenue, Pensnett Trading Estate, Kingswinford, West Midlands, DY6 7US, United Kingdom

SUMMARY

This paper considers factors affecting performance and efficiency in HVAC units and embarks on the development of a simple spreadsheet-based tool to predict performance, facilitating optimisation of inlet conditions. This tool is developed through benchmarking a ventilation unit with a 315mm inlet spigot, comprising an enclosure or rectangular cross section and a single backward curved centrifugal EC motorised impeller, of known performance characteristics, mounted on an inclined bulkhead.

INTRODUCTION

Keywords

DSa – Diameter of centrifugal impeller.

UVU – Uni-directional ventilation unit; A ventilation unit comprising one motorised impeller, with or without air treatment.

r/min – Revolutions per minute; unit of measure for rotational speed.

3D – Three dimensional

CFD – Computational fluid dynamics.

BEP – Best efficiency point; the duty point of the fan or ventilation unit at the highest point on fan efficiency curve.

BVU – Bi-directional ventilation unit; A ventilation unit comprising of two motorised impellers, with air volumes travelling in two directions, with or without heat recovery cross over or air treatment.

EC – Electronically Commutated; refers to a brushless, permanent magnet motor, an AC voltage is converted to DC voltage using integrated electronics to provide control and improved efficiency.

Background to the problem

Legislation introduced to increase the efficiency of energy-using products, such as fans and ventilation units offered to market presents ever greater challenges in the development of products that meet both minimum efficiency requirements and the needs and expectations of customers. Performance and efficiency of a ventilation unit can be improved though careful consideration of the path the conveyed air takes through a housing or enclosure, including customer connections at the inlet and outlet, and suitably guiding the air to reduce internal pressure losses.

A ventilation unit as defined by the EU Commission [1] "…means an electricity driven appliance equipped with at least one impeller, one motor and a casing and intend to replace utilised air by outdoor air in a building or a part of a building". These ventilation units can be further subcategorised into unidirectional (one impeller within an enclosure) either extracting air from or supplying air to a space; and bidirectional (two impellers within an enclosure) simultaneously supplying air to and extracting air from a space, often with a means of heat recovery.

Typically the impeller used in UVUs and BVUs is either of centrifugal or mixed flow type, and the motor is either an external rotor motor built into the impeller (motorised impeller), or a squirrel cage induction motor with a shaft mounted in the impeller. Major manufacturers of these types of impeller will produce fan characteristic curves stating pressure development vs airflow volume, as a free running impeller (without a casing or enclosure).

Some will also advise minimum enclosure clearances as a function of impeller diameter, in order to reduce the impact on stated free running performance. In practice, maintaining recommended clearances is not always possible; and a compromise needs to be made between the overall size of the unit and the resulting performance envelope under sub-optimal conditions.

With the advent, and wider adoption within industry of electronically commutated (EC) motors, and higher efficiency AC motors for use in Europe as mandated in EU2019/1781 (and before that EC640/2009), further gains in motor efficiency are likely to have reached a point of diminishing returns whereby the investment required, both pecuniary and in time effort, would be an order of magnitude higher than the potential improvement. Instead, how these motors are applied to applications such as fans and ventilation units comes into greater focus. Within these applications, efficiency of the impeller and how the combined motor and impeller are housed, are major contributors to the overall fan or ventilation units' efficiency and the energy consumed; to which separate legislation applies EU327/2011 & EU1253/2014.

This paper will consider factors affecting performance and efficiency in HVAC units and the development of a simple tool to predict performance, facilitating optimisation of inlet conditions. A potential benefit of this tool would be the reduction in physical prototype and computational fluid dynamics (CFD) iterations required in the design of new product ranges, allowing for quicker development cycles.

Motorised impellers (those that have motors built into the impeller), in particular EC types, will generally have been designed by Original Equipment Manufacturers (OEM) to comply with the latest efficiency tier legislation for a given performance envelope. As a result the last remaining tool available to a manufacturer of a fan or ventilation unit utilising these motorised impellers is in designing an enclosure or housing to suit the broader requirements of a completed product, whilst keeping to a minimum the internal pressure losses attributed to the enclosure.

Consequently, this research project, in looking at optimising inlet conditions to reduce internal losses, and development of a simplified tool to predict performance, has potential to be of interest to not only the author's employer, but the wider HVAC community.

RESEARCH DEFINITION

Practical problem/issue

In the development of ventilation units, ideal inlet and outlet conditions cannot always be achieved because of size and system constraints - such as any requirements for filters, heaters, sensors, attenuation materials etc, either built-in or positioned at the inlet or outlet of a unit. An option to lessen the impact on the airflow and electrical performance of the overall unit, is to improve the air path geometry reducing the losses attributed to the enclosure.

Physical performance test iterations to optimise air path geometry are time consuming and require significant materials. They are also prone to environmental factors such as human error or judgement when taking instrument readings.

Optimisation of air path geometry through CFD reduces the number of physical test iterations and introduces a greater control over the test environment; however, this requires a certain level of machine capability, operator skill and understanding to sufficiently set up and review results.

The practical problem is that the current product development process needs to speed up to increase throughput. Existing basic spreadsheet-based tools for performance prediction lack the required level of accuracy and variable options to significantly reduce the number of physical performance test iterations. This research project will seek to improve or redevelop the existing spreadsheet-based tools with a greater level of accuracy and functionality.

Existing relevant knowledge (review of literature)

Bayomi *et al.* [2] studied the effect of inlet straighteners of differing geometry on three types of centrifugal impeller, radial, backward curved, forward curved. The research used experimental investigation to measure the performance characteristics, comparing a corrugated zig-zag straightener fitted parallel to the direction of airflow, to a circular bundle of plastic straws also parallel to the direction of flow. The authors identified that the type of straightener geometry had different effects on performance depending on the type of impeller they were being used with. Radial and backward curved impellers presented an increase in either airflow volume, efficiency or decrease in noise.

Corsini *et al.* [3] reviewed computational methods employed by industrial fan designers used in the development of impellers, fans and use of HVAC components within systems. The paper is of relevance to this project as it is the first presentation whereby the advantage and disadvantages of each computation method is discussed, with respects to the type of CFD study being undertaken and the level of accuracy required. The type of solver methodology to use differs for say impeller design, where predictions of blade-to-blade interaction on flow field are important; to the design of a ventilation system where the prediction of the system performance is dependent on the correct setup of inflow and outflow boundary conditions, and modelling the fan's impact on the system with known pressure and volume characteristics rather than impeller geometry. It is the latter methodology best suited to this project, where the effect of an enclosure on performance is to be examined.

A detailed study by Fukue *et al.* [4] focussed on predicting the performance of cooling fans within an electronic equipment enclosure, and the effectiveness of heat transfer considering airflow obstruction arising from the positioning of electronic components. Similar to the project the author is undertaking, Fukue *et al.* [4], made use of fans which were supplied by a third party with a defined pressure-volume curve. The paper is an example giving credibility to the findings of Corsini *et al.* [3] which highlighted that CFD studies concerned with analysis of the overall ventilation system would be best served using a placeholder 'fan model' with flow rate and pressure differential attributes, rather than impeller geometry. They found that the maximum flow rate in each condition was determined by the inlet area and would only reduce once the inlet area was below two times that of the fan flow area.

Liu *et al.* [5] looked at performance improvements of a centrifugal fan using CFD simulation methods. Here the interface between an inlet cone and an impeller was investigated and optimised to reduce leakage losses occurring at the interface between the inlet cone and the impeller. It was found that through altering the linkage profile between the inlet cone and the impeller shroud, losses could be reduced and an improvement to the performance and operating range of the fan. This paper is of interest to this research project as it is another example of the effect inlet geometry has on the air path, and subsequent performance. Similar to the work by Yan *et al.* [6] this paper also considers the velocity profile of air, and how changes to profiles can work to improve performance.

An investigation by Yan *et al.* [6] looked at inlet flow distortion in a centrifugal fan, comparing two kinds of inlet duct arrangement. Here a straight duct and separately a duct arranged at 90 degrees to the inlet nozzle were numerically analysed to determine the effect of inlet acceleration on flow distortion and the knock-on influence on fan performance. Having established that the straight duct presented as "axial inflow which means the inlet flow without distortion", the authors found that the duct with 90-degree bend on the fan inlet exhibited flow distortion attributable to a non-uniform velocity profile in the inlet cross section. The result being a reduction of overall fan performance; specifically 3.3 % on pressure development and 2.5 % on efficiency over the straight duct.

A study by Argyropoulos and Markatos [7] reviewed developments in the numerical modelling of turbulent flow, including the suitability of the differing turbulence models for particular applications. k-ε is the most commonly used of the two-equation models, with established behaviour pattern and recommended for analysing flow away from boundary walls and gross estimation of flow fields. k-ω is described as being more accurate for separated flow characteristics, and at boundary walls with wide pressure gradients.

A common theme from the literature reviews is the use of the k-epsilon turbulence model, which is primarily used in studies where better production of "*…the energy cascade of large-scale structures in the main flow…*" [3], are needed. i.e., where the study is not too concerned about the interface between the flow around the wall boundary conditions. This study is of practical use to this project as it sets out in some detail the steps required to setup a study in Solidworks Flow Simulation (SFS) and demonstrates the suitability of SFS for a comparison of designs, which in the case of this project will involve enclosure shape and inlet incident angles.

Aim, objectives, methods, tasks and deliverables

This research paper aims to improve the design and performance of rectangular and square cross section box type ventilation units – in particular looking to reduce the inherent enclosure pressure losses and improve the efficiency of the unit (reduced electrical demand); which will be achieved through a set of underpinning objectives:

1. Investigate the effect different internal air guide geometry and inlet/outlet air path incident angles have on internal pressure loss.
2. Create CFD model for ventilation unit with known physical performance tests, in order to generate the CFD environment for optimisation.
3. Using CFD optimise air guide geometry (size, shape and quantity) and incident angles to the benefit of airflow, within conventional manufacturing methods.
4. Develop a spreadsheet-based computational tool to investigate optimum variables and predict the performance of ventilation units of differing sizes/motor configurations, following the conclusion of objective 3 and the generated data.
5. Make recommendations for the application of the computational tool in certain scenarios.

The intention is that in developing a spreadsheet-based computational tool, the speed of future development can be increased through the reduction of physical prototype iterations, and the de-skilling of the process to carry out a performance prediction during development feasibility studies.

METHODOLOGY

Methods and techniques selected

A review of the current state of the art, and developments in the field of fan engineering was carried out, including the benchmarking of existing known designs and principles. Having established the perceived strengths and weaknesses of earlier work based on comparisons between designs and the effect on performance, the authors sought to design an appropriate experiment and sensitivity study for this project, forming a piece of quantitative research.

A 3D model of the selected ventilation unit was produced in the CAD environment, Solidworks. The variables were assigned to configurations within the 3D model, allowing for ease of adjustment during the study. The 3D model was loaded into CFD package, Solidworks Flow Simulation (as the subsequent optimisation was used for comparative purposes against the original model studied). Suitable mesh, boundary conditions and turbulence model were applied to the model and set against a constant pressure differential.

Results obtained from the CFD process were reviewed, and the studies repeated using an optimisation process. Optimisation involved a parametric study in which incremental changes to variables including inlet angle (represented by dimension ‘A’ in Figure 2).

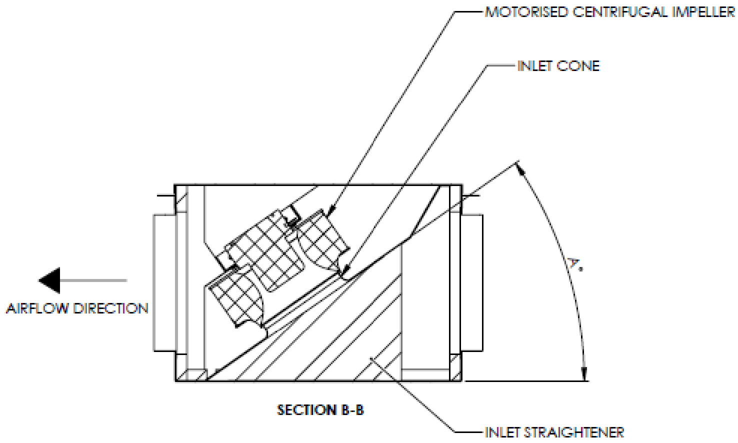

Figure 2: Example ventilation unit, with inclined inlet and straightener

Using the data collected from the simulations and the prior laboratory tests, comparisons were drawn between the results achieved for each configuration, and deviations noted. Any patterns emerging from the results are used to extrapolate between tested configurations.

A simple spreadsheet-based tool was created, using the data collected on the effect of changing each of the variables. The tool was constructed to take user inputs, in order to predict what the performance of a ‘new’ ventilation unit will be. These inputs consist of internal enclosure dimension, inlet angle, presence and type/shape of inlet guide and impeller size.

Justification

A Ø315 mm single fan was selected. Creating a potential opportunity for the research outcomes to influence future development on a higher sales-volume product whilst also having a large enough cross section to better analyse the flow. This unit is referred to as Model 1 (M1) and is cuboid (rectangular prism) in shape. It has a cross section of approx. 573 mm x 449 mm in the direction of airflow and a motorised impeller diameter of 310 mm. Using existing experimental data for Model

1 (M1), three duty points were plotted on the pressure-volume curve, as detailed in Table 1. These were used as the target points for the initial CFD benchmarking exercise, having been chosen to give the best representation of the full pressure-volume curve.

Table 1: Volume points chosen for benchmark CFD studies

Duty Point Reference	**Volume flowrate, m^3/s**	**Static Pressure, Pa**
BEP	0.350	425
75 % peak pressure	0.242	538
75 % peak airflow	0.495	230

This paper investigates the effect of enclosure size, and incident angle on the airflow performance of a ventilation unit using a motorised impeller with known 'free running' fan characteristics (pressure-volume). Since this research is concerned with the flow moving through an enclosure, and not in the development of an impeller from scratch, a rotating region or "fan model" approach to the CFD model was used with fan characteristics populated, in keeping with findings of [3]. Whilst there are several turbulence models which could have been used in the CFD environment, the one chosen for the studies was k-epsilon turbulence model as, unlike k-omega, it is better suited to main flow analysis and provides a good compromise on hardware requirements [7]. An environmental static pressure value was be applied to the inlet of the test duct model, a fan model region designated within the ventilation unit model and a static pressure target at the outlet of the test duct model. An automatic mesh size was used initially in the global domain, with a refined mesh in the region of the inlet to the motorised impeller.

The model consisted of the ventilation unit and an inlet and outlet duct replicating the ISO5801 setup used for the original laboratory performance test. Once good agreement between the CFD results and the three duty points was achieved, a level of confidence in the CFD environment to move on to the next stage of the research existed. The next stage involved the variation of the ventilation unit's cross-sectional area from a maximum of 1.8 times the motorised impeller diameter, in both the x and y axis, as recommended by some motorised impeller manufacturers.

RESULTS

Data collected

Several 3D fan unit models were generated with the configurations detailed in Annex B (Table 2) and a benchmark study was initiated to validate the CFD environment against previously obtained experimental data. The three fan duty points in Table 1, were selected from the benchmark unit to provide a representative performance cross section for plotting a fan curve, and for the setting of volume flow (airflow) rate in the CFD model.

Each of the CFD models were analysed, with the internal volume flow rate (m^3/s) and static pressure (Pa) at inlet and outlet of the ventilation unit recorded at incremental environmental pressure (Pa) differentials, to simulate the loading on the fan. For each CFD model, these pressure readings were compared to the free running impeller only performance, for a given air volume, and a pressure loss derived. Figure 3 shows an example of the CFD pressure cut plots obtained during the studies, and Figure 4 presents the resulting pressure loss curves for each CFM model configuration.

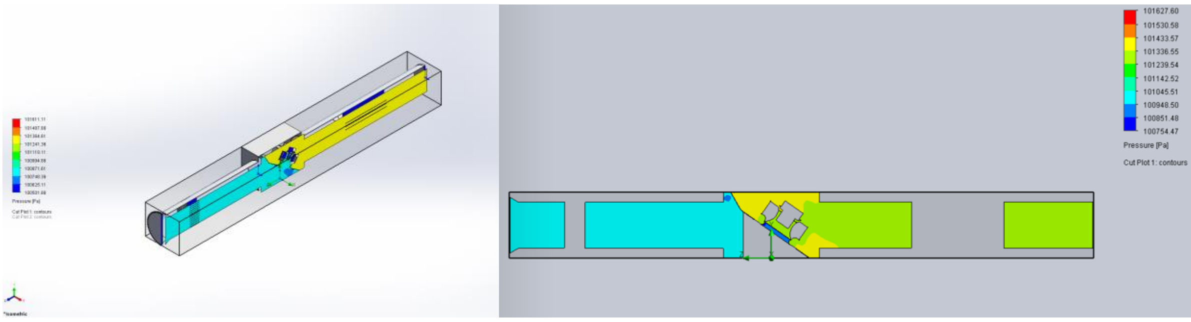

Figure 3: M1-M6 enclosure pressure loss curves plotted on graph

In addition to the CFD activities, an experimental test was carried out to confirm the airflow performance of the motorised impeller running without an enclosure. The test was conducted using a plenum chamber designed according to BS EN ISO 5801:2017, where the test item is mounted to the outside of the chamber, Figure 7, drawing air from within. The results from this experimental test were used to create a fan profile within Solidworks Flow Simulation and applied to the impeller model. [Annex A]

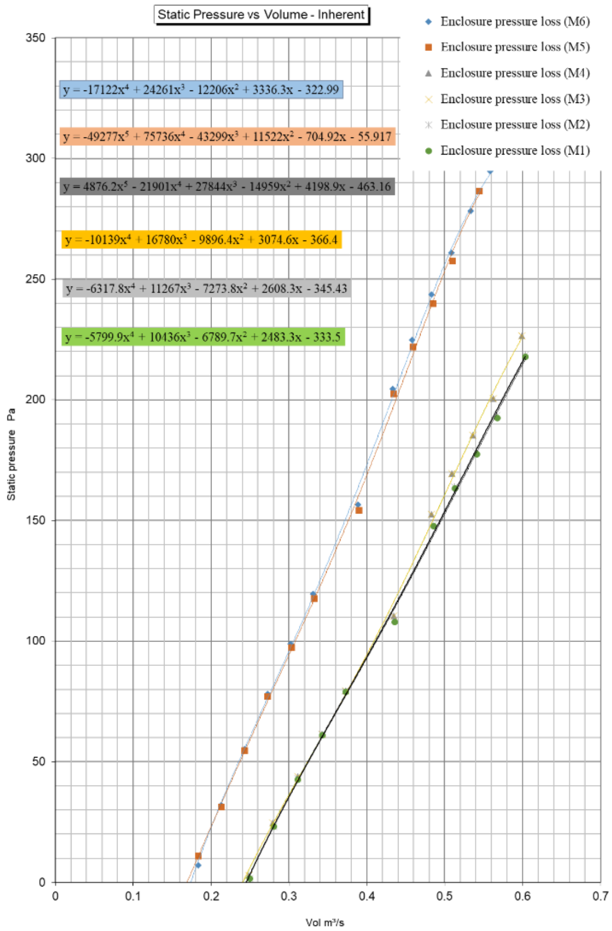

Figure 4: M1-M6 enclosure pressure loss curves plotted on graph

A review of past experimental data, Figure 5, carried out at the authors' employer yielded information on the effect of varying the inlet incident angle. In Figure 5, the blue curve shows the airflow performance of a fan with an inlet bulkhead incident angle of 0°. The black curve shows airflow performance of a fan with an inlet bulkhead incident angle of 35°, using the same motorised impeller and ventilation unit cross section. The results of this pre-existing experimental data, does correlate with the differences we see between the angled bulkhead in CFD models M1-M4, and that of the vertical bulkheads used in CFD models M5 & M6 (Figure 4).

With the CFD and real-world experimental comparison completed a parametric test was conducted using the M1 model as the base. This parametric study was set up to change the incident angle of the bulkhead and evaluate how, while maintaining enclosure size, adjusting the impeller mounting angle impacts the performance of the unit. The test would start with the impeller mounted at 0^{o} to the horizontal plane and increase the angle by 5^{o} for each run of the test, with a range from 0^{o} to 40^{o}, (Table 2). The resulting geometries, Figure 9, show the impeller maintaining a consistent height within the enclosure but moving slightly closer to the outlet side of the unit as the angle increases.

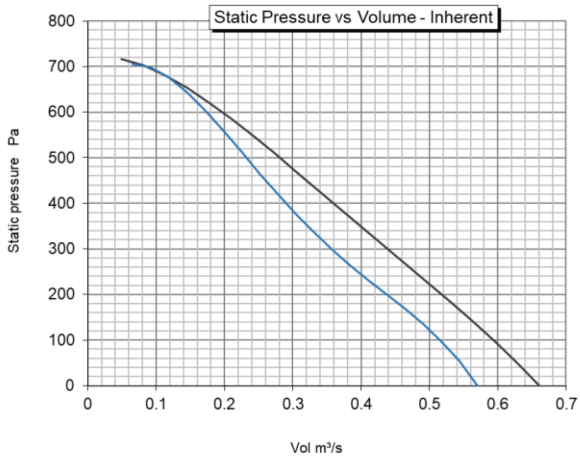

Figure 5: Graph showing airflow performance curves of one ventilation unit using two different motorised impeller mounting arrangements

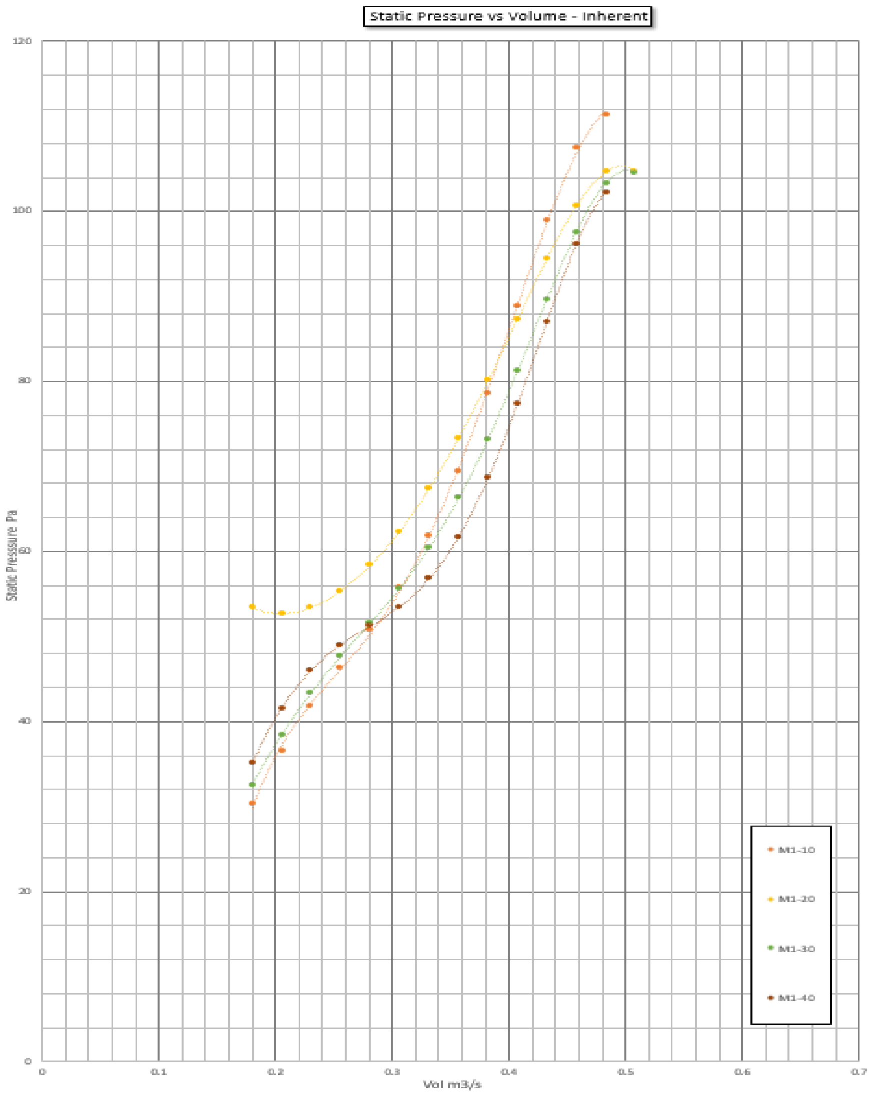

Figure 6: M1-10-40 enclosure pressure loss curves plotted on graph

Figure 6, shows the pressure loss across the unit for an angled bulkhead of 10°, 20°, 30° and 40°, for clarity the results for all test have not been included. The results are not easily distinguished, with areas of overlap at certain volume and pressure readings. Whilst there is not a clear "best option" emerging from the parametric CFD study, unlike the real world experimental test data between 35^o and vertical bulkhead (0°) which has a very clear difference in pressure loss, these results which span roughly the same delta θ, do not show a distinct difference in pressure loss value. Although a noticeable variation in gradient, between M1-10 (10°) and that of M1-40 (40°) is present, whereby M1-10 exhibits a steep gradient, indicating a higher rate of pressure loss over a given volume; and M1-40 which has a shallower gradient, indicating a lower rate of pressure loss for a given volume – which is in line with expectations.

CONCLUSIONS

Conclusions about the objectives and research aims

The results met some expectations in that the models using an angled bulkhead M1 to M4 generally presented with a lower value for enclosure pressure losses, when compared to those with a vertical inlet bulkhead M5-M6. What was not expected were the findings from this study that increasing the cross-sectional area of the enclosure had a negative impact; it is certainly not a scenario the author has encountered during real-world experimental testing in prior development projects and is an area in which further work could be carried out.

During this project, a greater understanding of the impact of enclosure design on the performance of a UVU has been gained. Opportunities exist to improve existing designs through the introduction of angled or inclined bulkheads, without too much investment in tools or machinery. The enclosure loss calculator has as a result of this project been updated to allow for incident angles to be entered, although somewhat limited in scope currently as cannot vary both angle and enclosure cross section together.

Further work

Having reached an end point in the research project, it is clear that there are opportunities for improvement and in further work being carried out. The close correlation of airflow performance curves generated through the CFD studies suggests that the minimum and maximum limits applied to the variables investigated were narrow in range. It is therefore suggested that subsequent research looks to widen the minimum and maximum limits of the variables, in particular enclosure cross-section. It would also be interesting to investigate the effect of enclosure length on the performance of a ventilation unit, and also spend time reviewing the electrical characteristic of the ventilation units with variation in size.

Additional future research projects could extend the scope to include BVUs and twin motorised impeller-based units in a run and standby configuration.

BIBLIOGRAPHY

[1] Commission Regulation, '(EU) No 1253/2014 of 7 July 2014 - Implementing Directive 2009/125/EC of the European Parliament and of the Council with regard to ecodesign requirements for ventilation units', p. 19, **2014**.

[2] N. N. Bayomi, A. Abdel Hafiz, A. M. Osman, 'Effect of inlet straighteners on centrifugal fan performance', *Energy Conversion and Management*, vol. 47, no. 18–19, pp. 3307–3318, Nov. **2006**, doi: 10.1016/j.enconman.2006.01.003.

[3] A. Corsini, G. Delibra, A. G. Sheard, 'A Critical Review of Computational Methods and Their Application in Industrial Fan Design', *ISRN Mechanical Engineering*, vol. 2013, pp. 1–20, Nov. **2013**, doi: 10.1155/2013/625175.

[4] T. Fukue, M. Ishizuka, S. Nakagawa, T. Hatakeyama, K. Koizumi, 'Model for predicting performance of cooling fans for thermal design of electronic equipment (Modeling and evaluation of effects from electronic enclosure and inlet sizes)', *Heat Trans. Asian Res.*, vol. 40, no. 4, pp. 369–386, Jun. **2011**, doi: 10.1002/htj.20347.

[5] X. Liu, Q. Dang, G. Xi, 'Performance Improvement of Centrifugal Fan by Using CFD', *Engineering Applications of Computational Fluid Mechanics*, vol. 2, no. 2, pp. 130–140, Jan. **2008**, doi: 10.1080/19942060.2008.11015216.

[6] W. Yan, D. Quanlin, X. Liu, 'Numerical investigation on the effect of inlet acceleration on inlet flow distortion in a centrifugal fan', presented at the Second International Conference on Mechanics, Materials and Structural Engineering (ICMMSE 2017), Beijing, China, **2017**. doi: 10.2991/icmmse-17.2017.55.

[7] C. D. Argyropoulos, N. C. Markatos, 'Recent advances on the numerical modelling of turbulent flows', *Applied Mathematical Modelling*, vol. 39, no. 2, pp. 693–732, Jan. **2015**, doi: 10.1016/j.apm.2014.07.001.

ANNEX A

Figure 7: Experimental test setup using plenum chamber

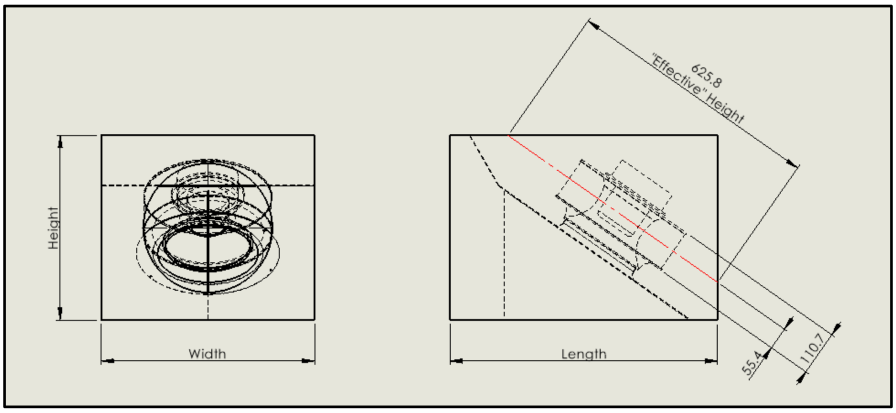

Figure 8: Drawing of M1 highlighting "effective" height on inclined plane

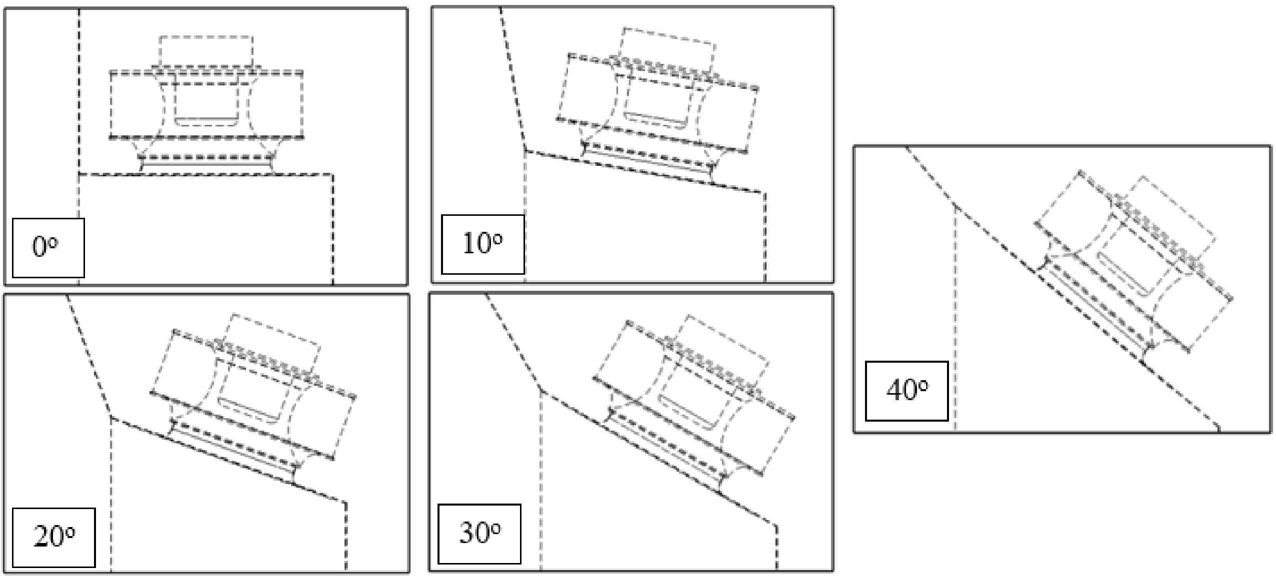

Figure 9: Bulkhead Angle Visualisation

ANNEX B

Table 2: CFD model matrix

CFD reference	Model description	Motorised impeller diameter (DSa) [mm]	Bulkhead angle [°]	Internal unit dimensions (normal to inlet) [mm]			Inlet straightener	Effective height [mm]	Effective impeller outlet cross-sectional area (at impeller midplane) [m²]
				Width	Height	Length	Centre		
M1	Benchmark case (Model 1)	310	35	520	449	652	1	625.8	0.325
M2	Benchmark case, without inlet guide	310	35	520	449	652	0	625.8	0.325
M3	Angled bulkhead, 1.8x DSa	310	35	558	558	652	1	743.8	0.415
M4	Angled bulkhead, 1.8x DSa, without inlet guide	310	35	558	558	652	0	743.8	0.415
M5	Vertical bulkhead	310	90	520	449	652	0	520	0.270
M6	Vertical bulkhead, 1.8x DSa - height & width	310	90	558	558	652	0	558	0.311
M1-0	Benchmark case (Model 1) with 0° Bulkhead Angle	310	0	520	449	652	1	531.3	0.276
M1-5	Benchmark case (Model 1) with 5° Bulkhead Angle	310	5	520	449	652	1	526.3	0.274
M1-10	Benchmark case (Model 1) with 10° Bulkhead Angle	310	10	520	449	652	1	528.9	0.275
M1-15	Benchmark case (Model 1) with 15° Bulkhead Angle	310	15	520	449	652	1	540.3	0.281
M1-20	Benchmark case (Model 1) with 20° Bulkhead Angle	310	20	520	449	652	1	563.2	0.293
M1-25	Benchmark case (Model 1) with 25° Bulkhead Angle	310	25	520	449	652	1	603.2	0.314
M1-30	Benchmark case (Model 1) with 30° Bulkhead Angle	310	30	520	449	652	1	597.2	0311
M1-35	Benchmark case (Model 1) with 35° Bulkhead Angle	310	35	520	449	652	1	566.8	0.295
M1-40	Benchmark case (Model 1) with 40° Bulkhead Angle	310	40	520	449	652	1	559.3	0.291