World Academy of Science, Engineering and Technology
International Journal of Electrical and Computer Engineering
Vol:4, No:12, 2010

Simulation of the Airflow Characteristic inside a Hard Disk Drive by Applying a Computational Fluid Dynamics Software

Chanchal Saha, Huynh Trung Luong, M. H. Aziz, and Tharinan Rattanalert

Open Science Index, Electrical and Computer Engineering Vol:4, No:12, 2010 publications.waset.org/13601/pdf

Abstract**—**Now-a-days, numbers of simulation software are being used all over the world to solve Computational Fluid Dynamics (CFD) related problems. In this present study, a commercial CFD simulation software namely STAR-CCM+ is applied to analyze the airflow characteristics inside a 2.5" hard disk drive. Each step of the software is described adequately to obtain the output and the data are verified with the theories to justify the robustness of the simulation outcome. This study gives an insight about the accuracy level of the CFD simulation software to compute CFD related problems although it largely depends upon the computer speed. Also this study will open avenues for further research.

Keywords**—**Computational fluid dynamics, Hard disk drive, Meshing, Recirculation filter, and Filter physics parameter.

I. Introduction

COMPUTATIONAL fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems those govern with fluid flows. Computers are used to perform millions of calculations required to simulate the interactions of liquids and gases with surfaces defined by boundary conditions. According to John D. [1], the physical aspects of any fluid flows are governed by the following three fundamental principles: 1. Mass is conserved, 2. F=ma (Newton's second law), and 3. Energy is conserved. These fundamental principles can be expressed in terms of mathematical equations which are the general form of partial differential equations. In addition, CFD is the art of replacing the governing partial differential equations of fluid flow with numbers, and advancing these numbers in space or time to obtain a final numerical description of the complete flow field of interest. The instrument which has allowed a large growth of CFD is the high-speed digital computer. Therefore, advances in CFD, and its application to solve more complicated and sophisticated problems, are strongly related to the advancement of computer hardware.

The role of CFD in engineering predictions has become so strong that today it is considered as a new 'third dimension' in fluid dynamics. Pure experiment and pure theory are the classical cases which are considered as other two dimensions. According to John D.'s [1] opinion, CFD supports and compliments both pure theory and experiment. In addition, with the dawn of high-speed digital computer, CFD will be considered as a third dimension with equal stature and significance of other two dimensions. From the basic research of engineering design, it has occupied the permanent place in all aspects of fluid dynamics.

In various engineering application areas, its physical configurations can be rather complex (e.g., aircraft, submarines, surface ships, hydro- and aero-propulsion devices and components, electromagnetic/ radar/antennae, etc.). The numerical solutions of these usually result in problem sizes that are extremely large and must be broken up into smaller pieces and then distributed in many processors working in parallel. Nowadays, a commercial simulation software (e.g., STAR-CCM+, Fluent and so on) are capable to handle the complexity governs with problems related to CFD. However, it requires high speed computer to ensure the accuracy and speed of the solutions.

CFD simulation software can be applied in a wide range of applications across almost all industrial sectors such as aerospace and defense, biomedical, automotive, buildings, chemicals, electronics, energy, environmental, oil and gas, marine, turbo-machinery, and so on. In this study, an airflow characteristic inside a 2.5" hard disk drive (HDD) is analyzed by STAR-CCM+ simulation software to conceptualize the approach of CFD software to obtain a solution.

II. Hard Disk Drive Model

This study investigates a 2.5" notebook HDD whose length, width, and height are 96, 66, and 12.5 mm respectively, and its unoccupied space volume is 23783 mm^3. It has two disk plates, diameter of each plate is 65 mm, and the distance between them is 1.94 mm. The length, width and height of the filer medium are 15, 7.65, and 2.05 mm respectively, and area of the filter inlet and outlet interfaces are 64.579, and 56.191 mm^2 correspondingly. Figure 1 shows the CAD model of the disk drive.

Chanchal Saha is with the Asian Institute of Technology, Bangkok, CO 12120 Thailand (phone: +66-87-3349454; e-mail: saha@ait.ac.th).

Huynh Trung Luong is with the Asian Institute of Technology, Bangkok, CO 12120 Thailand (e-mail: luong@ait.ac.th).

M. H. Aziz is with the Asian Institute of Technology, Bangkok, CO 12120 Thailand (e-mail: muhammad.haris.aziz@ait.ac.th).

Tharinan Rattanalert is with the Asian Institute of Technology, Bangkok, CO 12120 Thailand (e-mail: ying_at_first@hotmail.com).

Open Science Index, Electrical and Computer Engineering Vol:4, No:12, 2010 publications.waset.org/13601/pdf

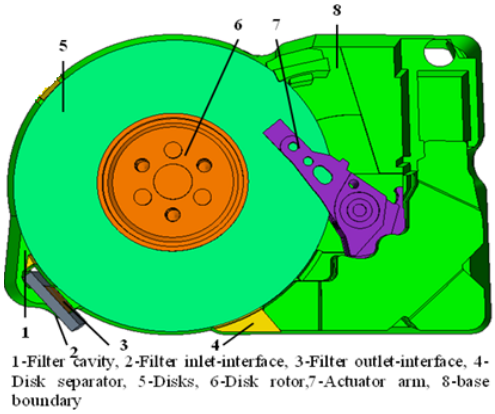


Fig. 1 Hard disk drive CAD model

III. Methodology

The following flow-chart shows necessary steps required to setup the model for data acquisition and visualization of airflow model. The whole processes are described as follows:

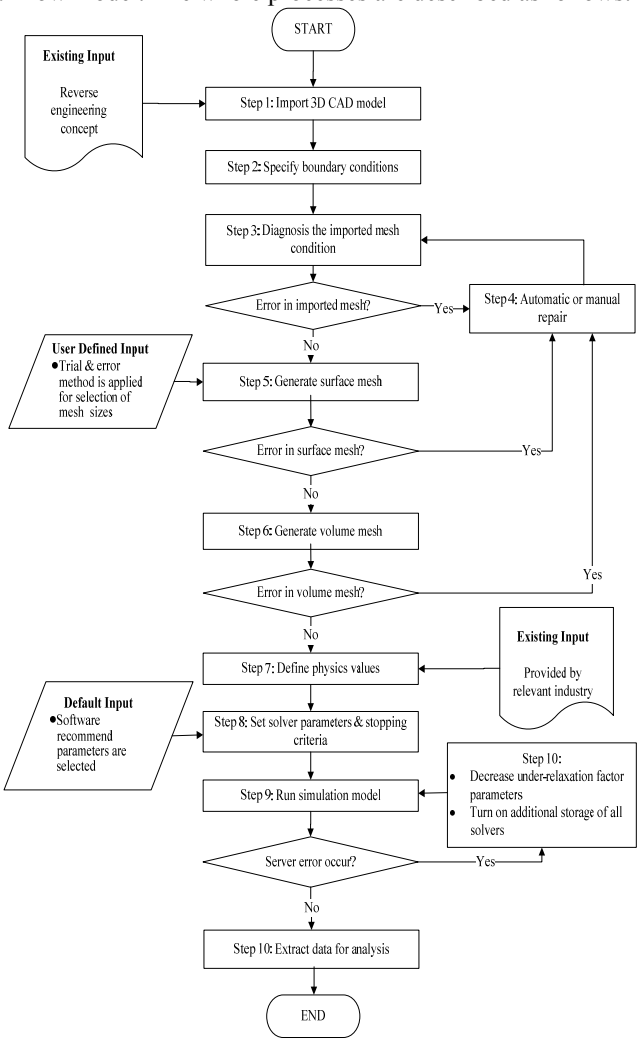


Fig. 2 Flowchart to setup model for data acquisition

A. Import

A CAD file is imported and it is found that disk drive is divided into nine regions namely body 1 to body 9. It is observed that except 'Body 3' other parts can be discarded because all the essential parts are already combined with 'Body 3'. In addition, it is unwise to develop a model with a large amount of cells those are not really essential. The main reason is that, there is a threat of dealing with large number of bad cells. Moreover, it will take longer time to run the simulation model to reach the convergences with larger number of cells. Considering all these facts, other regions are discarded except 'Body 3' and divided into two regions, namely 'Disk drive' and 'Filter' before checking the mesh factor scale. These regions hold all the essentials parts of the disk drive those are needed to carry out the airflow model. The regions have 75128 faces whereas the number of close proximity, and poor quality faces are 2859, and 3688 respectively. At this level, it is effective to define boundary conditions before doing the repair of surface. The following figure 3, the left image represents the imported disk drive model.



Fig. 3 Imported HDD and close proximity faces (orange in color) model respectively

B. Boundary conditions

Before doing the repair of the mesh, the regions are split into required numbers of boundaries. In this case, the 'Disk drive' is split into seven boundaries namely Top disk, Disk separator, Base disk, Actuator arm, Surface boundary, Disk rotor, and Base boundary, and the 'Filter' region has only one boundary namely 'Filter'. Several techniques are applied here to split a region into boundaries. Feature curve, angle, patch techniques are applied here to split the region. First of all, new feature curve is created to mark the edges of the whole 'Disk drive' with an angle of 31 degree. It will mark all the edges those are greater than or equal to 31 degree to capture the outer shape of each and every parts of the disk drive. Next, split by patch separate the upper portion of the disk drive from other parts and it is renamed as 'Surface boundary'. Later on, the existing feature curve is edited by selecting single edges to mark the actuator arm, disks, disk separator and rotor separately and create feature curves for each of them. At the end, the 'Disk drive' region is split into Actuator arm, Base disk, Top disk, Disk rotor, Disk separator and Base boundary according to the feature curve angle created for each boundary. Splitting is necessary to represent the effect of different parameters over the parts of the body individually as well as collectively. The disk drive is considered as a closed

Open Science Index, Electrical and Computer Engineering Vol:4, No:12, 2010 publications.waset.org/13601/pdf

volume. So, all the boundaries are considered as wall that mean there is no inlet or outlet for the air to come in or go out. At this point, interfaces have been created between 'Disk drive' and 'Filter' region which is namely 'inlet' and 'outlet'. It provides a connection between the two regions where the air can pass from one region to another region. 'Filter' is considered as porous region and the two interfaces as porous baffle boundary.

C. Repair

After defining the boundary, repair of the CAD model is necessary before generating the mesh of the model. It is not possible to generate a volume mesh if there is any defect specially pierced faces or non-manifold edges and vertices over the surface mesh. In order to get a defect free surface, mesh diagnostic report helps to find out the defective area over the surface. After detecting the error faces, it can be repaired either automatically or manually. In this case, auto repair tool is well enough to repair the defective surface areas. In right side image in figure 3, orange color represents the close proximity faces.

D. Meshing

The CAD model is now ready for meshing. The meshing is divided into two parts. First of all, surface meshing and next, volume meshing. To get an accurate output, the mesh model is very important. Diagnostic report is run to know the quality of meshing after completing each meshing. The size of the mesh should be chosen in such a way that it can capture small changes over the disk surface. It is known that all the governing equations are run in each mesh cell and it gives a single output. This output is then considered as an input for the surrounding meshes and in this way the whole model runs to obtain a solution. Under this condition, if the cell size is too big then it will not able to present an accurate result of the changes that take place within a very small area. On the other hand, if the mesh cell size is relatively small then it requires more CPU time and memory size as with the decrease of cell size it needs more number of iterations to solve the solver equations in each cell. So, it is mandatory to find out a suitable mesh size which is a tradeoff between the CPU configuration and goal of this model. The surface meshing should be run on different mesh base size to find out a suitable size based on the mesh diagnostic result and condition of mesh surface after re-mesh.

TABLE I
LOCAL SPECIFICATION OF MESH PARAMETERS

Boundaries of the regions	Base size (in)	Target size (in)	Minimum size (in)	Prism layer stretching	Prism layer thickness (in)
Actuator arm	0.1	0.1	0.02	1.1	0.02
Base boundary	0.1	0.1	0.02	1.2	0.033
Base disk	0.1	0.1	0.02	1.1	0.02
Disk rotor	0.1	0.1	0.025	1.2	0.033
Top disk	0.1	0.1	0.02	1.1	0.02
Surface boundary	0.1	0.1	0.05	0	0
Filter	0.1	0.02	0.01	0	0

In this study, base size is considered as 0.1 inch with target size 100% of base size and minimum value of the cell size is specified as 25% of the base value for the whole region which is selected as a global parameter. Later on, mesh size is set locally for each boundary considering its importance to save the CPU iteration time.

For volume meshing, polyhedral mesher is chosen with prism layer mesher. Volume meshes fill-up the volume inside the surface mesh, so that solver equations can work inside that area. Polyhedral mesh selected here for volume meshing, however, a comparative study can be done to find out which mesh can be preferred between polyhedral and trimmed mesh. In addition, prism layer is selected to take care of the corner of each surface to obtain better solution as it needs to be ensured that prism layer is placed nicely at the corner of each boundary. Number of prism layer is 2, prism layer thickness 0.02 inches and layer stretching is set at 1.1 as a global parameter. The following table 1 presents the local parameters set for the surface and volume meshing for the boundary regions. Figure 4 shows the surface, volume and prism layer meshes of the disk drive respectively.

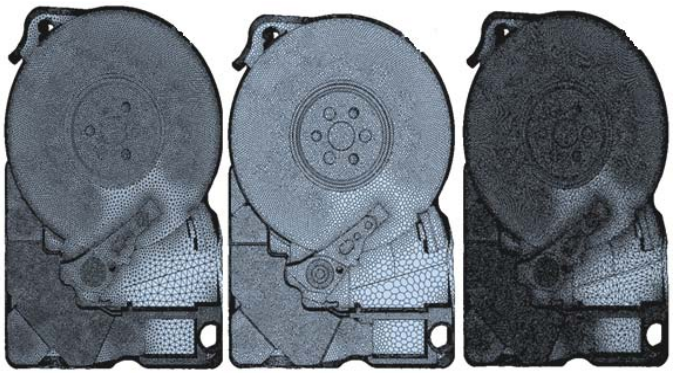


Fig. 4 Surface, volume and prism layer meshes respectively

E. Physics

After constructing the mesh model, next step is to form the physics model. To obtain the desired goal, physics model should be specified accordingly. Auto select physics model is activated to allow the software to choose few more options those are suitable for this model. In this study, the parameters those are specified in physics model are shown in table 2 and their descriptions are given below.

TABLE II
PHYSICS MODEL PARAMETERS

Gas	Three dimensional
Steady	Segregated flow
Constant density	Stationary
Turbulent	Reynolds averaged Navier stokes
K-epsilon turbulence	Realizable k-epsilon two layer
Two-layer all y+ wall treatment	Cell quality remediation

The material model is responsible for managing various thermodynamic and transport properties relevant to that material and physical processes being modeled in the continuum. 'Gas' is selected as the fluid material inside the disk drive and its material properties is considered as air. In

Open Science Index, Electrical and Computer Engineering Vol:4, No:12, 2010 publications.waset.org/13601/pdf

this study, the density and viscosity of air are considered as 1.184 kg/m^3 and 1.855×10^{-5} Pa-s respectively.

The primary function of the space model is to provide methods for computing and accessing mesh metrics such as cell volume, face area and centroid, cell and face indexes, and skewness angle. Three space models are available in Star-CCM+ software namely two-dimensional, three-dimensional and axisymmetric model. In this study, three-dimensional model is selected as it is designed to work on 'Three-dimensional' meshes, and should only be activated if the mesh is indeed three-dimensional.

The primary function of the time models in Star-CCM+ is to provide solvers that control the iteration and unsteady time-stepping. Implicit and explicit unsteady time models use with implicit and explicit solver models respectively. In this case, 'Steady' time model is selected for steady-state calculations.

The 'Segregated flow' model is selected in this study as it solves the flow equations (one for each component of velocity, and one for pressure) in a segregated, or uncoupled, manner and its algorithm uses less memory than the coupled. The linkage between the momentum and continuity equations is achieved with a predictor-corrector approach. Moreover, the segregated Flow model is suitable for incompressible or mildly compressible flows.

The equation of state model is used to compute the density and the density derivatives with respect to temperature and pressure. 'Constant density' model is selected in this case as the density is invariant throughout the whole continuum of gas material.

To correctly predict the fluid behaviors, it is necessary to model motion. The 'Stationary' model is used here that does not involve any kind of motion. A rotation axis condition, however, can be added to any boundary region for which the stationary model is active.

The viscous regime deals with the types of flow in fluid medium. In this disk model, the flow often changes its direction with the rotation of disk. For this reason, 'Turbulent' model is selected as a viscous regime as it deals with the flow in a state of continuous instability, exhibiting irregular, small-scale, high-frequency fluctuations in both space and time. 'Raynold-averaged Navier-stokes (RANS)' is the default turbulence model for this software.

'K-epsilon turbulence' models provide a good compromise between robustness, computational cost and accuracy. They are generally well suited to industrial-type applications that contain complex recirculation, with or without heat transfer. For this reason, this model is selected with RANS model. The 'Realizable two-layer k-epsilon' model combines the Realizable k-epsilon model with the two-layer approach. The coefficients in the models are identical, but model gains the added flexibility of an 'All y+ wall treatment'. The all- y^+ wall treatment is a hybrid treatment that attempts to emulate the high-y^+ wall treatment for coarse meshes and the low-y^+ wall treatment for fine meshes. It is also formulated with the desirable characteristic of producing reasonable answers for meshes of intermediate resolution (that is, when the wall-cell centroid falls within the buffer region of the boundary layer). This is the recommended wall treatment provided by the software.

The 'Cell quality remediation' model is enabled in an effort to obtain solutions on a poor-quality mesh. This model attempts to identify poor-quality cells based on a set of predefined criteria, such as skewness angle exceeding a certain threshold. Once these cells and their neighbors have been marked, the computed gradients in these cells are modified in such a way as to improve the robustness of the solution.

The porous baffle parameters that are chosen for filter and its interfaces are as follows:

- Porosity: 0.4
- Porous inertial resistance (α): 28.842 kg/m^4
- Porous viscous resistance (β): 158.13 kg/m^3-s

A Cartesian coordinate system namely 'Disk coordinate' is set at the middle of the disk (e.g., 6.76, 3.67, 0) cm from the origin to allow the disks and rotor to rotate at the anti-clockwise direction on that point around the z-axis at a constant speed of 5400 rotation per minute (RPM). Another Cartesian coordinate system namely 'Filter coordinate' is set at one corner (8.55, 6.74, and 12) cm of the filter from the origin for better presentation of the output. Initial condition and physics values are kept as a default value of the soft-ware as it supports the objective of this study.

F. Solver setting and stopping criterion

As the model is a segregated flow type, the solver setting can be kept as its default value and under-relaxation factor parameters for each solver setting can be changed between 0 and 1, as it necessary to stabilize the iterative process of steady state solutions. Lower value of the under-relaxation factor is an excellent choice; however it causes a very slow convergence. The stopping criterion is set for 500 iterations to achieve an acceptable level (less than three-order-magnitude drop) of convergences.

G. Run model

After completing all the steps mentioned above the model is now ready to run for collecting data. While running the model, residual plot should be observed carefully. Whether there is any server error or the residual plot is diverging, in that case the following few steps can be followed and run the model again for seeking output.

- Decrease the value of under-relaxation factors of all solvers than its previous value.
- Turn on the additional storage of all solvers.

After completing the iteration process the model is ready for data extraction.

IV. RESULT ANALYSIS

Fig. 5 shows that continuity plot magnitude drops more than a three-order magnitude. Whereas, X, Y, Z-momentum, Turbulent dissipation rate (Tdr) and Turbulent kinetic energy (Tke) drop more than a four-order magnitude. It is an obvious indication of the solution converges.

Figure 6 shows scalar velocity field plots over the disk surface respectively. Scalar velocity plot shows that velocity magnitude is increasing gradually from inner diameter towards outer diameter of the disk ranges from 0 to 18.401 meter per second (m/s).

Open Science Index, Electrical and Computer Engineering Vol:4, No:12, 2010 publications.waset.org/13601/pdf

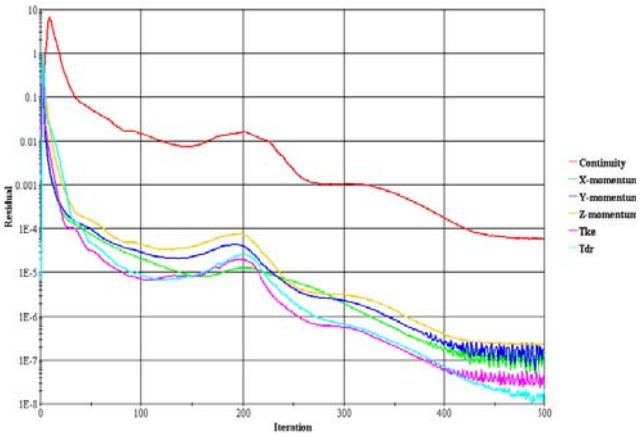


Fig. 5 Residuals monitor plot of the airflow model

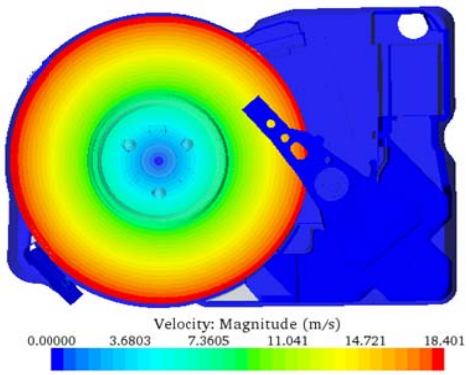


Fig. 6 Scalar velocity field magnitude over disk drive

In the following figure 7, the direction of the airflow can be analyzed. It can be seen that velocity is low near the filter interfaces and direction of the airflow changes dramatically due to narrow space near filter cavity.

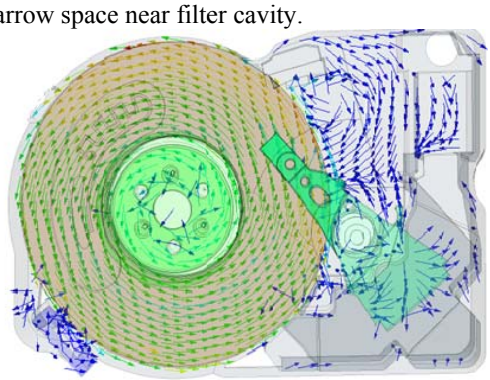


Fig.7 Airflow direction over disk drive

In figure 8, blue and yellow color lines represent the change of velocity magnitude with respect to the position over both disks for the plane section derived across X-and Y-axis respectively. In left hand plot, the Y-normal plane section is showing the linear increase of velocity magnitude from 8 m/s to 18.404 m/s for top disk and in right hand plot, the increment is 7.8 cm to 18.404 cm for base disk along the radius from 1.41cm to 3.25 cm. On the other hand, X-normal plane section characterizes the change of velocity along the X-axis direction.

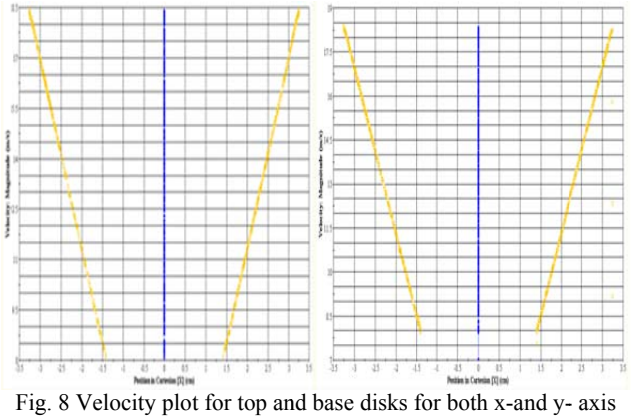


Fig. 8 Velocity plot for top and base disks for both x-and y- axis section respectively

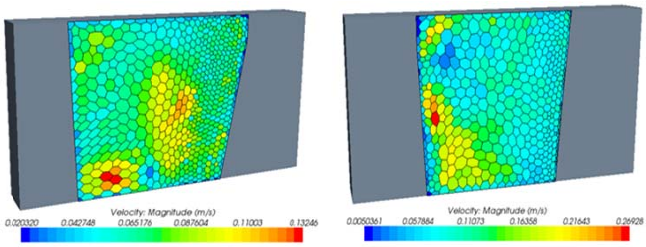


Fig. 9 Scalar velocity plot at filter inlet and outlet interfaces respectively

The left image in Figure 9 represents the scalar velocity plots for filter inlet interface where the value changes between 0.0203 to 0.1325 m/s. It shows the velocity magnitude on inlet interface which has higher velocity at the lower and middle part of the interface. The right hand plot shows the change in velocity between 0.00503 to 0.26926 m/s along the filter outlet interface. It can be observed that at the lower left part of the outlet interface has the maximum velocity.

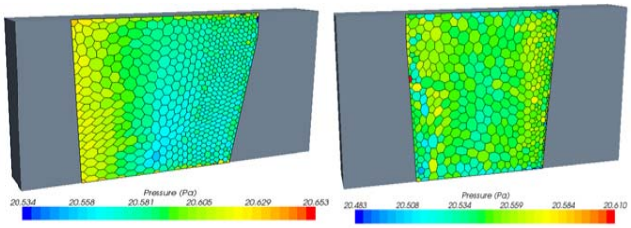


Fig. 10 Pressure plot at filter inlet and outlet interfaces

In the above figure 10, pressure magnitude varies at inlet interface between 20.534 to 20.653 Pa whereas at outlet interface, it fluctuates between 20.483 to 20.610 Pa.

TABLE III
VELOCITY AND PRESSURE DATA ACROSS THE FILTER INTERFACES

α:28.842 β:158.13 Porosity:0.4	Velocity (m/s)			Pressure (Pa)		
	Max	Min	Avg	Max	Min	Avg
Filter inlet	0.1325	0.02	0.0715	20.653	20.534	20.586
Filter outlet	0.2693	0.005	0.1	20.61	20.483	20.547

From table 3, it can be seen that there is an increase in velocity magnitudes and at the same time decrease in pressure

magnitudes from inlet interface to outlet interface due to the presence of porous baffle filter region.

V. Verification

In this study, the maximum velocity over the disk is obtained as 18.403 m/s whereas the diameter of the disk is 6.5 cm and disk rotation speed is 5400 rpm. After putting the value of disk rotation speed and disk diameter in the theoretical equation $v = 2\pi fr$, it is found that the velocity is 18.4 m/s which are similar with the simulation output. The table 4 shows a comparison among the maximum velocity obtained by the researchers from different disk drive model research. From the table, it can be observed that the results of this study are comparable with the previous study outcomes.

From the filter medium airflow model, it is found that across filter interfaces there are pressure drops and velocity lifts at the same position over the interfaces. It can be explained by Bernoulli's equations [2]. Therefore, it supports the claim that at a specific point total energy should be equal (e.g., drop of pressure with the increase of velocity and vice versa).

TABLE IV
COMPARISON TABLE OF DISK DRIVE ROTATION SPEED

Study	Disk size (cm)	Rotation speed (rpm)	Maximum velocity (m/s)
(Ding & Kumar, 2007)	4.8	4200	8.9
(Ding & Kumar, 2007)	9.5	7200	18.1
(Song, Murali, & Ng, 2004)	10	5400	26.4
Current study	6.5	5400	18.403
Current study	6.5	7200	24.54

VI. Conclusion

This study gives an insight about each of step of specific software to solve a computational fluid dynamics related problem. Despite of the massive development of the simulation software and computer hardware, it demands a lot of patience from the software user to get a nearly accurate result as the whole process governs under trial and error basis. Therefore, from the analysis, it is found that velocity magnitude is linearly increasing form center to outer diameter of the disk drive. Furthermore, velocity and pressure magnitudes are higher at the bottom and left corner of the filter interfaces respectively. In addition, velocity magnitude inside filter cavity is very low compared to the rotating disks velocity.

Therefore, the present thesis forms a foundation for further investigations by varying disk rotational speed with filter sizes and positions inside disk drive. In addition, the study can be further extended by injecting particles of different sizes and densities from various locations inside disk drive to observe the contaminated particles behavior which can help to design a better recirculation filter for disk drive.

References

[1] John D. Anderson Jr., Jooris Degroote Gerad Degrez, Erik Dick, Roger Grundmann, and Jan Vierendeels, "*Computional Fluid Dynamics: An Introduction,*" Berlin Heidelberg: Springer-Verlag.

[2] Paul Deed, and Paul Edmondson, "*What is Bernoulli's Equation and Its Applications,*" www.hubpages.com: http://hubpages.com/hub/What-is-Bernoullis-Equation-and-its-applications, Retrieved April 18, 2010.

[3] CD-adapco, "*STAR-CCM+ version 4.02.007user's guide,Singapore,*" Singapore: CD-adapco,2009.

[4] William Ding, and Manish Kumar," *Hard disk drives recirculation filter design by simulating particle and operation parameters effects on particle trajectories,*" Bloomington, Donaldson Co. Inc, 2007.

[5] Hanbing Song, Damodaran Murali, and Quock Y. Ng, "*Simualtion of flow field and particle trajectories in hard disk drive enclosures,*" Massachusetts: DSpace@MIT, 2004.

Open Science Index, Electrical and Computer Engineering Vol:4, No:12, 2010 publications.waset.org/13601/pdf