A computational study of the effects of airflow, temperature and particles trajectories in a small form factor hard disk drive enclosure

Goh, Elson Hong Joo

2008

Goh, E. H. J. (2008). A computational study of the effects of airflow, temperature and particles trajectories in a small form factor hard disk drive enclosure. Master's thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/41739>

<https://doi.org/10.32657/10356/41739>

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

**A COMPUTATIONAL STUDY OF THE EFFECTS OF
AIRFLOW, TEMPERATURE AND PARTICLES
TRAJECTORIES IN A SMALL FORM FACTOR HARD
DISK DRIVE ENCLOSURE**

**ELSON, GOH HONG JOO
SCHOOL OF MECHANICAL AND AEROSPACE
ENGINEERING
2008**

**A COMPUTATIONAL STUDY OF THE EFFECTS OF
AIRFLOW, TEMPERATURE AND PARTICLES
TRAJECTORIES IN A SMALL FORM FACTOR HARD
DISK DRIVE ENCLOSURE**

ELSON, GOH HONG JOO

SCHOOL OF MECHANICAL AND AEROSPACE ENGINEERING

A thesis submitted to the Nanyang Technological University
in fulfillment of the requirement for the degree of
Master of Engineering

2008

TABLE OF CONTENTS

Abstract	i
Acknowledgement	ii
List of Figures	iii
1. INTRODUCTION	1
1.1 Background	1
1.2 Literature review	8
1.3 Objective and scope of this research	23
1.4 Layout of this thesis	24
2. COMPUTER AIDED GEOMETRIC MODELING OF HARD DISK DRIVE ENCLOSURES AND MESH GENERATION	25
2.1 Overview on modeling process of the HDD enclosure with artifact boundaries	26
2.2 HDD assembly model mesh generation	29
2.3 Software interface error	45
2.4 Automatic volume mesh scheme	47
2.5 Mesh generation at HDI	51
2.6 Mesh adaption	58
3. COMPUTATIONAL MODELLING OF FLOW FIELD CHARACTERISTICS AND PARTICLE TRANSPORT IN HARD DISK DRIVE ENCLOSURES	62
3.1 Flow models	62
3.2 Discrete Phase Model (DPM) for computing trajectories	74
3.3 Fine Particle Model (FPM) for air-field representation	81
4. RESULTS AND DISCUSSIONS	83
4.1 Computed airflow characteristics in a basic HDD enclosure	84
4.2 Computed airflow characteristics in a geometrically exact HDD enclosure	102
4.3 Particle transport in a geometrically exact HDD enclosure	124
4.4 Fine Particle Model (FPM) on particles interaction	142
4.5 Local mesh refinement using solution-adaptive method	147
4.6 Flow improvement with wall shaping	152
5. CONCLUSION AND RECOMMENDATIONS FOR FURTHER WORK	155
5.1 Conclusions	155
5.2 Recommendations for further work	158
REFERENCES	159

Abstract

This thesis presents the result of a numerical investigation on the airflow characteristics and particle trajectories inside a small form factor hard disk drive (HDD) enclosure. A simple model and a geometrically exact model of the HDD enclosure are meshed using unstructured tetrahedral elements. Using a commercial flow-solver, the incompressible Navier-Stokes Equations are solved for the two computational domains. The superior results, in term of resolution and accuracy extracted from the detailed model are compared to the simpler model.

Using the computed results, insight on the airflow characteristics in the vicinity of the slider are obtained. The effect of a geometrical true arm model on the global flow field is also addressed for typical disk operation conditions. The computed airflow patterns are then used to predict particle trajectories in the vicinity of head/disk interface (HDI), the knowledge of which has relevance for tribological aspects connected with the HDI region.

The effects of gravity and thermal gradients on the airflow characteristics within the HDD enclosure are also considered. The understanding of particle trajectories and the interactions with the forces in the flow domain provide useful guidelines for HDD design and re-circulation filter placement. The effect of thermophoresis force on varying size particles is assessed by defining a heat source wall within the domain and recomputed. An alternative model for predicting particles distribution based on the Eulerian Frame of reference is also used to assess the state of particle contamination in the HDD enclosure.

Acknowledgement

This work has been supported by a joint project between Seagate Technology International and Nanyang Technological University (NTU). The work has been carried out in the Centre for Advanced Numerical Engineering Simulation (CANES) Laboratory of the NTU. The author would like to express his sincere appreciation and gratitude to Associate Professor Murali Damodaran for his invaluable advice and guidance throughout this research work. The author also acknowledges useful discussions with Dr. Q. Y. Ng and on the particle transport model with Dr. S. Ali for this work.

List of Figures

Figure 1.1: History of increasing recording density	4
Figure 1.2: Components inside the HDD enclosure	4
Figure 1.3: Head Stack Assembly.	5
Figure 1.4: Head Gimbal Assembly (HGA) showing Head Disk Interface	5
Figure 1.5: (a) Schematic drawing showing the orientation of colored light sources and the video camera.	10
(b) Definition of the two angles defining the orientation of flakes	10
Figure 1.6: Sketch of the particle-generating set-up.	13
Figure 1.7: Slider/disk interaction in HDD	13
Figure 1.8: Experimental Set-up	14
Figure 1.9: Schematics of particle sampling point	14
Figure 1.10: Model used for the validation study. Cross-section of the shrouded co-rotating disks (Wu and Chen [24])	21
Figure 1.11: Radial Profile of the circumferential mean velocity [23]	21
Figure 1.12: 3D Rail topography of slider with recesses on surface	21
Figure 1.13: Coordinate system	22
Figure 1.14: Pressure profile under a flat slider	22
Figure 1.15: Effects of the initial velocity on the path of particles: 1- $U_{p0} = 0.4$; 2- $U_{p0} = 0.5$; 3- $U_{p0} = 0.6$; and 4- $U_{p0} = 0.8$ 20	22
Figure 2.1: Workflow diagram illustrating the major steps in CFD setup.	28
Figure 2.2: Geometrical model of HDD directly imported from CAD system	28
Figure 2.3: Rendered view of arm assembly from modeling software	32
Figure 2.4: Perspective view of Head Gimbal Assembly (HGA)	32
Figure 2.5: Unstructured triangular surface mesh on the HGA	32
Figure 2.6: Surface mesh on slider with size function on mesh growth rate	33
Figure 2.7: Dimple and slider (a) CAD model (b) Surface mesh	33

Figure 2.8: Flexure and limiter	34
Figure 2.9: Lofting the arm suspension	34
Figure 2.10: Unmerged edge on the arm suspension	34
Figure 2.11: Surface mesh on rotor-hub wall with gradual refinement towards disk (Disk is omitted for clarity of presentation).	35
Figure 2.12: Close up at suspension vicinity. Mesh is concentrated in the vicinity of slider	35
Figure 2.13: Unstructured grid displayed on plane 1 between top disk surface and slider	34
Figure 2.14: Perspective view of (a) load/unload ramp (b) Ramp assembly in HDD	40
Figure 2.15: Surface meshes on ramp	40
Figure 2.16: Breather filter modeling	40
Figure 2.17: Re-circulation filter holder	41
Figure 2.18: Exploded layers of re-circulation filter	41
Figure 2.19: Rendered views in different angles (a) constructed in <i>Rhino</i> [43] (b) Extracted from Seagate isometric drawing	42
Figure 2.20: Rendered view of top cover (a) Extracted from Seagate isometric drawing (b) Constructed in <i>Rhino</i> [43]	42
Figure 2.21: Rendered view of motorbase (a) Extracted from Seagate isometric drawing (b) Constructed in <i>Rhino</i> [43]	43
Figure 2.22: The interior wall features of HDD enclosure generated by <i>Rhino</i> [43]	43
Figure 2.23: CAD Models of (a) HDD mechanical structure (b) Fluid region inside HDD enclosure	44
Figure 2.24: Interception of a cone and block volume with coincident control Point	44
Figure 2.25: Surface mesh on complex HDD enclosure walls	44
Figure 2.26: Wire frame view of dimple on suspension in (a) <i>Rhinoceros</i> 3D (b) <i>Gambit</i>	46

Figure 2.27: Suspension in (a) Wire frame view (b) mesh quality inspection mode on skewness	46
Figure 2.28: Modified edge on arm suspension to eliminate acute mesh angle	46
Figure 2.29: Volume mesh built from (a) a single HDD enclosure CAD volume (b) split or multiple CAD volumes.	49
Figure 2.30: Splitting of single CAD volume.	49
Figure 2.31: Simple T-Grid meshing scheme (a) for unmeshed cube (b) Simple T-grid mesh	50
Figure 2.32: Pyramidal elements adjacent to the pre-meshed suspension edge	50
Figure 2.33: Velocity contours of split volume: No kinks at boundaries and flows are continuous	50
Figure 2.34: Nodes distribution in the Head Disk Interface (HDI) based on (a) evenly distribution node (b) Unevenly distributed nodes.	54
Figure 2.35:(a) Top view of split disk area (b) mesh distribution on disk surface with smooth cell growth from slider edges.	54
Figure 2.36: (a) Surface mesh on arm (b) Size function attached on slider surface	56
Figure 2.37: Schematic diagram illustrating a dirty edge	56
Figure 2.38: (a) Real HDD volume in Gambit (d) Close up on lift tab	56
Figure 2.39: Figure 2.38: Nodes distribution on lift tab with (a) Coarse face meshes from automated meshing scheme (b) vertices removal from edge (c) Nodes of fine intervals assigned to edges (d) Good quality unstructured triangular surface mesh	57
Figure 2.40: Types of face element with (a) Quadrilateral (b) Triangular	57
Figure 2.41: Section of volume mesh across plane 1 (Between top disk surface and slider): The volume mesh size is limited to 0.7 interval to enable cell smoothness	58
Figure 2.42: Example of a hanging node	60
Figure 2.43: Grid distribution taken along the parallel axis of the arm at the mid width (a) before adaption (b) after adaption.	60
Figure 2.44: 510903 marked cells for adaption	60

Figure 2.45: Grids (a) before adaption (b) after adaption, solution adaption based on relative large velocity variation	61
Figure 3.1: Contours of computed cell Reynolds Numbers	65
Figure 3.2: Contours of computed cell Reynolds Numbers (a) Laminar flow (b) RSM, turbulent flow	66
Figure 3.3: Control volume used to illustrate discretization of a scalar transport equation	67
Figure 3.4: Brownian motion	79
Figure 3.5: Possible boundary condition upon particles impact	80
Figure 4.1: Convergence plots based pm (a) maximum velocity (b) maximum total pressure	91
Figure 4.2: Arm and disk assembly showing disk rotation direction and the position of the arm at inner diameter	92
Figure 4.3: Z-X view on arm assembly, showing heat dissipation from the motor-hub wall	92
Figure 4.4: Visualization (a) Plane 1: Interval between top slider and disk (b) Plane 2: Interval between bottom slider and disk.	93
Figure 4.5: Visualization (a) Plane 3: Y-Z plane of hub midsection (b) Plane 4: X-Z plane of hub midsection	93
Figure 4.6: Visualization (a) Plane 5: Midsection along the arm axis (b) Plane 6: Midsection along the ramp axis	93
Figure 4.7: Top views of simple model and arm at (a) ID (b) MD (c) OD	94
Figure 4.8: Snapshots of laminar field in the drive. Plot of velocity magnitudes on plane 1 (a) Arm at ID (b) Arm at MD and (c) Arm at OD.	96
Figure 4.9: Snapshots of laminar field in the drive. Plot of static pressure on plane 1 (a) Arm at ID (b) Arm at MD and (c) Arm at OD.	97
Figure 4.10: Contours plot of (a) Velocity magnitude and (b) Static (gauge) pressure at plane 3 and 4.	100
Figure 4.11: Perspective view of arm at ID for (a) Velocity magnitude contour (b) streamlines	101

Figure 4.13: Streamline trace without disk	101
Figure 4.14: Snapshots of laminar field in the exact drive model. Plot of velocity magnitudes vectors on plane 1 (a) Arm at ID (b) Arm at MD and (c) Arm at OD.	103
Figure 4.15: Snapshots of laminar field in the exact drive model. Plot of total pressure contour on plane 1 (a) Arm at ID (b) Arm at OD and (c) Arm at OD.	104
Figure 4.16: Static pressure (gauge) contour under the HDI plane	105
Figure 4.17: Velocity magnitude in vectors form illustrating the flow phenomenon around the slider.	105
Figure 4.18: Pressure gradient across plane 1 with arm positioned in ID (a) $dP-dX$ (b) $dP-dY$	106
Figure 4.19: Temperature contour on plane 3 and 4	106
Figure 4.20: Velocity vector plot on a (a) normal plane cutting through the edge of breather filter (b) close up view at the entry to breather filter	111
Figure 4.21: Velocity profile from top disk surface to top cover, couette flow	112
Figure 4.22: Streamlines tracked from the upstream of arm for 500 steps	112
Figure 4.23: (a) Velocity vector plot along plane 5 (b) slider surface pressure plot overlays with velocity plot on plane 5.	113
Figure 4.24: Negative gauge pressure across (a) plane 1 (b) plane 2	114
Figure 4.25: Streamlines tracked from filter exit	115
Figure 4.26: Streamlines (a) around slider (b) from upstream of slider	115
Figure 4.27: Re-circulation filter cutting diagonally across filter holder	118
Figure 4.28: Convergence history of mass flow rate on re-circulation filter	118
Figure 4.29: Velocity vectors plot on plane 1 (a) Re-circulation filter (b) No re-circulation filter.	119
Figure 4.30: Arbitrary line defines created through the filter for plotting flow variables	119
Figure 4.31: X-Y plot for velocity magnitude along arbitrary line comparing between filter effect and without filter effect	120

Figure 4.32: Static pressure (gauge) contour across plane 1 (a) Re-circulation filter (b) No Re-circulation filter, for pressure range between -2 to 4 Pa.	120
Figure 4.33: X-Y plot for velocity magnitude along arbitrary line comparing between filter effect and without filter effect	121
Figure 4.34: Velocity magnitude contour for (a) Laminar flow (b) RSM Turbulent flow	123
Figure 4.35: Pressure contour for (a) Laminar flow (b) RSM Turbulent flow	123
Figure 4.36: Particles release from leeward of slider wall.	124
Figure 4.37: Total (gauge) pressure contours on plane 1 for case (a) with heat source (b) no heat source, plot between -10 to 10 Pa.	130
Figure 4.38: Temperature gradient at plane (a) 3 plane and (b) 4 cutting the motor axis.	130
Figure 4.39: Alumina trajectories in the vicinity of HDI	131
Figure 4.40: Variation of the particle height in the z-direction vs distance along the particle trajectory for particles released from wall 1 of top slider.	131
Figure 4.41: Highly refined surface meshes on disk with a size growth of 1.08 from slider edges.	131
Figure 4.42: Alumina particles trajectories released from top slider	132
Figure 4.43: Alumina particles trajectories released from bottom slider	132
Figure 4.44: Trajectories of particles released from the four side walls of (a) top slider (b) bottom slider neglecting effects of gravity on particle transport.	133
Figure 4.45: Trajectories of particles released from the four side walls of (a) top slider (b) bottom slider including effects of gravity (b) on particle transport.	134

Figure 4.46: (a) Total (gauge) pressure contours, and (b) Velocity vectors on plane 5 cutting through the middle line drawn along the length of the arm and parallel to the z-axis	135
Figure 4.47: Computed gauge pressure contours (a) on z-y plane 3 and (b) on z-x plane 4, cutting through the rotor hub.	136
Figure 4.48: (a) Alumina path lines releasing from the top slider without heat force (b) Alumina path lines for top slider with heat effect	137
Figure 4.49: Particle trajectories of POM (a) around the disk (b) released from the ramp slope.	138
Figure 4.50: (a) Velocity vectors along plane 6 (b) Static (gauge) pressure along plane 6	139
Figure 4.51: Display planes showing the leading edge (plane 7) and trailing edge (plane 8) of the re-circulation filter	139
Figure 4.52: Velocity magnitude vectors (a) on plane 7 (b) close up at the leading edge of re-circulation filter.	140
Figure 4.53: Velocity magnitude vectors (a) on plane 8 (b) close up at the trailing edge of re-circulation filter.	141
Figure 4.54: (a) Wire frame top view of an 1-inch HDD (b) Arm suspension with slider as releasing source	144
Figure 4.55: Percentage of cells in the HDD enclosure populated with particles of given diameters at stead state	144
Figure 4.56: Particle concentration distribution due to effect of (a) diffusion (b) diffusion with heat force	145
Figure 4.57: The evolution of particle concentration on an iso-surface of particle diameter (a) 0.048 micron and (B) 0.0886 micron. Contours represent the particle concentrations.	146
Figure 4.58: Velocity contour with (a) no adapted mesh (b) adapted mesh.	149
Figure 4.59: Grid distribution at (a) 40 μm (b) 90 μm above disk.	149
Figure 4.60: Velocity contours at (a) 40 μm (b) 90 μm above disk before Mesh adaption.	150

Figure 4.61: Velocity contours at (a) 40 μm (b) 90 μm above disk with mesh adaption	150
Figure 4.62: Vorticity Magnitude contour taken from plane at 10 μm above disk (a) Before adaptive mesh (b) With adaptive mesh.	150
Figure 4.63: Vorticity Magnitude at cells zone (a) Left Top (b) right top (c) left bottom (d) right bottom	151
Figure 4.64: Arm & Ramp assembly (a) Arm superimpose with vorticity magnitude above 40,000 (1/s).	151
Figure 4.65: An air-guide incorporated onto breather filter (homogenous material) to guide streamlines to the outer-disk and minimize impingement of particles onto disk surface.	153
Figure 4.66: Top/bottom view of air-guide with breather filter.	153
Figure 4.67: Air- at bottom disk to guide particles away to outer disk	154
Figure 4.68: (a) Front end (ID) of air-guide with taper groove to capture low pressure flow. (b) Rear end (OD) of air-guide with enlarged opening to ease flow purging at higher pressure.	154
Figure 4.69: Particle trajectories of 0.1 μm spherical alumina with bottom air-guide. Alumina particles are guided outwardly.	154

CHAPTER 1

INTRODUCTION

1.1 Background

In recent years, hard disk drives (HDD) have spread in the field of consumer electronics (CE) from its dominating history in the field of Information Technology (IT). Hand-in-hand with a rapid increase in recording density has been the appearance of manufactured products with various “form factors” (i.e. size and format) and practical HDD applications ranging from stationary devices (like HDD recorders) to mobile devices are growing.

As the demand for portable digital music and video grows, mobile device manufacturers will need ample storage capacities. Small form factors HDD therefore provides a cost effective, high capacity storage solution for these device manufacturers compared to other storage alternatives. Rapid developments in the technology of miniaturization led to the shrinkage of many consumer electronics while constantly exhibiting improvements and novel performance characteristics. The need for high data storage capacity in consumer electronic devices continues to be one of the major forces driving the demand for ever-increasing miniaturization of hard-disk-drives (HDD).

Over its fifty years life span, the HDD has evolved from a 5MB storage device with 24 inches diameter disk platter, (about the size of a household refrigerator) to an 1-inch diameter disk platter. As a result of the increase in recording density, and boosted by cost reduction resulting from mass production, the proportion of CE devices equipped with HDDs has been increasing in recent years. The 1-inch small form factor HDD measures 40x30x5 mm, allows for more streamlined media players product designs and fits nicely into current clamshell and candy style mobile phones. With a storage capacity of up 12GB, the 1-inch HDD can be found in many *Apple iPod* and other MP3 players today.

The development of perpendicular recording as well as research on magnetic and optical combined recording have advanced over the last decade. In comparison with the surface recording density of the world’s first HDD, the present recording density has

been improved by about 65 million times. Accompany this dramatic leap in recording density, the miniaturization of the size of HDDs has continued up till the present. The smaller size form factors (1 inch) are finding a multitude of application as shown in Figure 1.1 [1]. Perpendicular recording technology that is replacing longitudinal recording on HDD combined with other enhancements should extend the useful life of the HDD for about another 14 to 15 years. At some point before year 2020, a major technology shift or revolution will be required unless a significant breakthrough in physics and material science can extend the HDD even further. Storage technologies or disk systems traditionally have been replaced on three-to-five-year cycles that should enable users to deploy several more iterations and generations of HDD-based storage before some new technology is defined and developed and products are ready for mission-critical deployment. Signs of a new major technology shift should be observed in about nine to ten years. However, between now and then, many smaller (on a relative scale) technology improvements and evolutionary enhancements will appear. The 1-inch hard disk drives (HDD) are revolutionary products and are excellent innovations of miniaturization. With the increase in sophisticated functions in small handheld devices, smaller form factor HDDs will inevitably replace the 3.5-inch HDD.

Figure 1.2 shows the schematic view of a typical 1-inch HDD enclosure showing all the various components and also delineating the space inside the HDD enclosure for the airflow induced by disk rotation to take place. The major components in a HDD enclosure are the base, arm, ramp, disk, re-circulation filter, breather filter, spindle motor, latch and magnet as indicated respectively in the figure. The base provides the platform for all internal components to be held together as a working unit. The arm is controlled to seek across the disk for the execution of the read and/or write operation on both sides of the magnetic disk. The disk is rigidly clamped to the motor-hub, and rotates at a specified rotating speed usually at 3600 rpm or 5400 rpm for the small form factor HDD, in the anti-clockwise direction. The ramp made from a polymer has a smooth surface that favors low frictional forces while the lift-tab makes contact with its surface bringing the arm away from disk surface or loading onto the data-track on disk. The breather filter adheres to the top cover by an adhesive surface. This filter contains a diffusion tube, connecting to an opening on the top cover to the inner volume

via a permeable mesh. The tube allows the exchange of air from the exterior of the HDD enclosure into the interior enclosure of the enclosure. The capillary exchange of air is very slow. This filter contains carbon that absorbs organic contaminants and aids in the removal of excessive moisture, thereby prevent large humidity changes in the drive that otherwise might result in condensation. The re-circulation filter provides an alternative filtration system in the HDD. It is ideally located in regions where large mass flow occurs and oriented with the largest cross-sectional area surface normal to the bulk flow direction thereby increasing possibilities of particle impingement onto the filter wall. The purpose of the latch is to retain the arm assembly when the drive is not operating by preventing it from bouncing onto the disk when the motor is not spinning and therefore without the air-bearing the slider will be damaged by contact with rough surface of disk.

During the start/stop operation of the HDD, the arm assembly would load/unload itself using the lift-tab from the ramp. The slider is attached to the suspension by swaging a ball bearing through holes on the suspension. The head and suspension assembly is termed the head-gimbal assembly (HGA) as shown in Figure 1.4. The HGAs are stacked together into a head-stack assembly (HSA), which is propelled across the disk surface by the actuator as shown in Figure 1.3. The actuators on most HDDs employ the voice coil mechanism. It functions very much like the voice coil in a loud speaker. It consists of a voice-coiled magnet (VCM), usually with a strong magnetic field in the 2000 ~ 8000 Gauss range. The actuator arm pivots about a bearing, which rotates the assembly setting the read/write head to seek from the inner disk to outer diameter during its functioning. Sending electric current to the actuator coil creates magnetic torque during start-up operation that controls the actuator arm movement.

Figure 1.1: History of Increasing Recording Density [1]

Figure 1.2: Components inside the HDD Enclosure

Figure 1.3: Head Stack Assembly

Figure 1.4: Head Gimbal Assembly (HGA) showing Head Disk Interface (HDI)

Figure 1.4 shows a view from the side of the slider and disk during a read/write operation. The Head Disk Interface (HDI) refers to the air film formed between the slider and the spinning disk when disk is spun and is in the range of a few nanometers. The HDI creates a repulsive air-bearing force that lifts the slider from the spinning disk surface at a constant fly-height, thereby enabling data transfer between these components.

While there are numerous benefits offered by miniaturization trends in HDD design, inevitable trade-offs in terms of manufacturability, such as dimension control result in huge investments in production for precision tools. Also the reduced space inside the small form factor HDD enclosure in which airflow is induced by the disk rotation increases the vulnerability of the components in the HDD enclosure to particle-induced damage. For the same sensitivity as defined by the ratio for air volume to the volume bounded by HDI, an 1-inch HDD enclosure will constitute a much smaller ratio and therefore higher tendency for foreign particles to get trapped in the HDI. This also underscores the importance for controlling the transport of particles at even very small length scales. One of the ways to minimize particle collision in the HDI is to analyze the impact of HDD enclosure wall shapes to airflow induced by disk rotation. Wall shaping is a possible approach to create a change in the global airflow distribution pattern inside the HDD enclosure. From the knowledge of the global flow characteristics, one can then gauge which regions inside the HDD enclosure need refined analysis. The information also provides guidance for the placement of filters. The understanding of the airflow distribution inside the HDD enclosure also aids in the understanding transport of particles of various random sizes within the HDD enclosure. This in turn provides useful information on the flow structures in the vicinity of the HDI where important tribological aspects could be addressed. HDD designers could make use of the insight to create more robust HDD read/write head slider designs that will divert particles away from the read/write head regions. An efficient and economic way to examine these effects on the design and prototyping of HDDs is to exploit modern computational fluid dynamics (CFD) software technology. The use of this technology overcomes the need for

simplifying the geometry or in making radical assumptions for modeling, thereby providing a reasonable prediction of these effects by solving exact mathematical models for airflow in the HDD in a complex HDD enclosure defined by exact geometrical shapes of all the components in the HDD.

Recent years have seen a surge in research activities focusing on various aspects of the HDD. The research on HDD is spurred by the motivation to increase the areal bit density of the read/write head so as to provide higher density recording, which is the main goal of many HDD manufacturers. To increase in the areal bit density, a lower flying height is regarded to achieve good stability control as well as a higher resolution in recording signals. With the reduction in HDI, smaller particles are highly susceptible of getting trapped in the HDI causing head damages. Many researchers who work on HDD use simple CAD representative models, paying little attention on the effect of HDD enclosure wall shapes especially on the HSA. Simplification has also the advantage that parametric studies could also be carried out with these setups economically. In the limit of suitable visualization tools to interpret the flow characteristics in such a small scale HDD, it would be less expensive and effectively to analysis the flow field computationally.

In this numerical study of the airflow induced by disk rotation and particle transport within the HDD enclosure, a three-dimensional geometric model which attempts to represent the exact industry standard 1-inch HDD is replicated with a geometric modeling tool. For example, the arm shown in Figure 1.3 has intricate geometrical protuberances and the fine detailed shapes of the slider, flexure, dimple and also its limiter, which will significantly affect the flow field in the vicinity of the HDI. If the arm is assumed to be devoid of all these fine details then the effect of the fine details on the arm on the flow can never be captured. Based on the geometric model of the HDD enclosure the free space inside the enclosure in which airflow is induced by disk rotation is discretized using a mesh generation tool to facilitate the numerical solution of the incompressible Navier-Stokes equations in the enclosure. The computed flow field inside the HDD is then used as a basis in conjunction with particle transport prediction models to predict particle trajectories in the enclosure.

1.2 Literature Review

The study of airflow characteristics in the HDD enclosure has received considerable attention over the years since the inception of HDD based storage technology. There have been significant experimental, theoretical and numerical research activities to study and analyze the airflow characteristics in the HDD over the past thirty years. The efforts aimed at investigating such flow characteristics can be generally classified into two main areas, namely, experimental techniques employing flow visualization (using wool tufts, smoke tracers, electric arc tracers and particle injection) and numerical/computational analysis employing modern computational fluid dynamics technology to solve the three-dimensional Navier-Stokes equations to extract complex flow physics in the HDD enclosure. A review of these activities is outlined in this section mainly to highlight the fundamental issues associated with an extremely complex device to establish the existing body of knowledge in this field.

1.2.1 Outline of Experimental Methods on HDD Enclosure Flowfields

The experimental work of Lennemann [2] was one of the earliest experimental investigations directly focused on disk drives. The author used model disks of diameter between 355.6 and 459 mm running at 710-600 rpm and used water and aluminum powder for flow visualization. Experiments were performed with and without a slider arm. Results demonstrated the existence of a central laminar core that is rotating slightly slower than the disk and a highly turbulence outer region. Kaneko *et al.* [3] performed similar flow visualization experiments to study the flow between disks with and without cylinder shroud. They observed a “laminar core” that extended from the hub to the mid-radius of the disks, followed by a “more turbulence outer region.” Abrahamson *et al.* [4] performed experiments using an acid-based indicator, Bromothymol Blue, in water. Disk speeds were varied from 5 to 50 rpm, the disk diameter was fixed at 112 cm. They observed three distinct regions in the flowfield comprising of a solid body inner region near the hub, an outer region dominated by counter rotating vortices and a boundary layer region near the shroud.” They reported that decreasing the Ekman number ($Ek = \nu/R^2\Omega$)

namely the ratio of viscous to Coriolis forces or increasing the axial spacing between the disks resulted in fewer vortical structures in the outer region and consequently greater overall mixing. Girard *et al.* [5] investigated the effect of an actuator-like rotary arm on the flow field in the drive, using water based flow visualization. Their main conclusions were related to the effect of the arm and the wake it creates.

Flow visualization techniques using laser-Doppler velocimetry techniques have looked into the solid body rotating of the flow between co-rotating disks [6]. Dijkstra *et al.* [7] used the finite different method to compute the flow between two finite rotating disks enclosed by a cylinder, and compared the prediction with experimental results by means of stereophotography. According to them, viscous effects are mainly confined to the thin boundary layers on the disk and cylinder surfaces at low Ekman numbers. More relevant to this scenario are experimental works that focus on the fluid flow between one rotating disk and a cylindrical enclosure. Zhou and Garner [8] used the Particle Image Velocimetry (PIV) technique to measure the planar flow field inside an enclosed cylindrical chamber containing a smooth rotating disk. They compared the experimental data obtained for two laminar cases near the transitional flow regime with the results obtained from a numerical model and found close agreement. Similarly, Escudier [9] who performed flow visualization on the bulk rotating fluid flow in the enclosure at low Reynolds number, showed that the flow in a cylindrical container induced by a rotating end wall is influenced by 2 parameters namely, the height-to-radius ratio and the rotational Reynolds number. Observations made using the laser-induced fluorescence technique are also presented for the steady swirling flow produced in a closed cylindrical container completely full of fluid (a glycerine/water mixture for the experiments reported here) by rotating one endwall.

As mentioned earlier, Thoroddsen *et al.* [10] presented a novel flow-visualization technique utilizing reflective flakes in combination with color illumination. Three differently colored colimated light beams directed from separate directions as shown in Figure 1.8(a) are used to illuminate the flow. Figure 1.5(a) shows the arrangement of the illuminating beams. The red and green light beams are arranged on the same vertical plane, lying parallel to the cylinder axis. This illuminates the flow symmetrically, giving information about the angle α as indicated in Figure 1.5(b). The test liquid contained

96% de-ionized water and 4% by volume of Kalliroscope AQ-1000 concentrate as formulated by Tirumkudulu *et al.* [11], which is a suspension of organic guanine concentration of flakes, which will limit the reflections to flakes in the vicinity of the glass surface. The orientation of the flakes in the complex three-dimensional flow is in general a complicated function of the local velocity gradient tensor, but can be calculated if the underlying velocity field is known. In complex flow fields, the distribution of flakes may, however, be arranged by the motion, thereby making the local intensity of reflection depend on both orientation and flakes concentration. The reflections will also be localized by the shallow focal depth of the optical lens. The color is, however, immune to the local number density of flakes inside the flow, making quantitative information possible.

Figure 1.5 (a) Schematic showing the orientation of colored light sources and the video camera.
(b) Definition of the two angles defining the orientation of the flakes.

This technique is demonstrated by visualizing the finer details of vortices in a Taylor-Couette device [12], which consisted of a brass inner cylinder and an outer cylinder made of 6mm thick glass to enable flow visualization. The 370mm gap between the inner and outer cylinders was filled with a fluid. The outer cylinder was fixed while the inner cylinder was rotated. This method is recommended to achieve better resolution of the flow structures during the visualization of airflow distribution in view of the small form factor HDD enclosure.

The experimental study of particle transport in the HDD enclosure involves the injection of particles in the domain of interest occupied by the working fluid. Most of these experimental results are categorized by the mean time before failure (MTBF), which describe the particle robustness based on head/media design. The higher particle robustness here relates to a longer failure time for the same numbers of particles injected into the HDD enclosure. M. Roy and J. L. Brand [13] devised a component-level test to study data erasure on both glass and aluminum media due to particles trapped between the head and the disk. Aerosol particles are generated using a commercially available collision nebulizer such as the BGI, Inc., Waltham, MA, model CN-24-1 as shown in Figure 1.6.

Aluminum oxide particles or florescent latex spheres are injected into an aqueous suspension inside the nebulizer reservoir. The nebulizer is filled with a suspension of alumina or florescent latex sphere and is driven by clean dry air (CDA) at a 10-25 PSI pressure to generate small liquid droplets, a percentage of which contain an individual particle of interest. The aerosol is then passed through a diffuser drier where the primary aerosol constituents, water droplets, evaporate leaving the encapsulated particles in the airflow. At the same time, the drier would reduce the relative humidity of the aerosol air to below 40 %. Subsequently, as the flow leaves the drier, the aerosol may be passed through a radioactive ionizer to reduce the electrostatic charge distribution developed during the atomization process to correspond to the Boltzmann equilibrium charge distribution [14]. After the particle has been generated, the aerosol stream used to carry the particles is split via a 4-way splitter. One stream is sent into the particle counter for size and characterization while another is used as the source for drive-level injection. The aerosol stream passes through a drier to remove the water aerosol. The ionizer and diluter are optional. The split flows are: the injection/deposition stream, a monitoring stream, and a pressure/flow equalization stream. Alumina particles are injected into the servo track writer head port as seen in Figure 1.6. The distribution of particles across the upper surfaces appears random and does not depend significantly on the injection location as determined by inspection of media from doped drives with optical surface

analyzer (OSA). This suggests that impingement immediately following exit from the aerosol output nozzle is negligible.

In addition, three main parameters are important for drive design feature. These are the dose time, aerosol density and hard drive assembly for which three different designs were injected. The total number of particles on the η^{th} surface (??) is the sum of three potential contributions i.e., $\eta_n^{\text{total}} = \eta_n^{\text{dynamic}} + \eta_n^{\text{settle}} + \eta_n^{\text{initial}}$, where η_n^{dynamic} is due to attachment of particles to the media while the aerosol is flowed through the drive, η_n^{settle} is that which settles following cessation of the aerosol flow and η_n^{initial} is that due to any initial contamination before injection. The aforementioned study provides insight on the type of boundary conditions to be imposed in the analysis for numerical simulation and serves as a good comparison based on a time-iterative transient set-up.

Lee *et al* [15] had investigated the effect of disk rotational speed on particles generated inside a HDD enclosure. Condensation nucleus counter (CNC) and particle size selector (PSS) had been used for this investigation. In addition, the particles were also examined by using SEM (scanning electron microscopes), AES (auger electron spectroscopy) and TOP-SIMS (time of flight-secondary ions mass spectrometry). It was observed that increasing disk rotational speed directly affected the particle generation by slider disk interaction. The particle rate generation increased rapidly at motor speed above 8000 rpm and the particles were generated from lubricant on the disk, coating layer of the disk and also slider surface. Figure 1.7 shows the slider disk interaction, which generates contamination particles and causes tribocharging phenomenon in the slider disk interface. The experimental set-up consist of clean dry air (CDA), particle measurement and analysis devices as shown in Figure 1.8.

Figure 1.6: Sketch of the particle-generating set-up.

Figure 1.7: Slider/disk interaction in HDD

Figure 1.8: Experimental Set-up

Figure 1.9: Schematics of particle sampling point.

Particles generated during HDD operation have been sampled by a probe (stainless steel) as shown in Figure 1.5, at flow rate of 0.3 or 1.4 L/min. The air that was sucked by the sampling probe was compensated by the same volume of clean air, filtered through a HEPA filter, to maintain 1 atm inside the drive. Particle size measurements were carried out with a CNC and one of its accessories, PSS. CNC has a low limit of 14 nm at 50% detection and is operated at the 1.4L/min sampling flow rate. PSS is a particle separating device that selectively removes small particles from an aerosol and passes large particles. It consists of a set of fine-mesh diffusion screens in a housing, which can be placed on the aerosol inlet of CNC. The motion of aerosol particle smaller than 0.2 μm in diameter is strongly affected by diffusion. Small particles with high diffusion coefficients rapidly diffuse to the wire screens where they are captured; larger particles with lower coefficients pass more readily through the screens. Particles extracted by the probe were captured with a membrane filter, which has pore size of 15 nm. This experiment investigated the self generate particles capability by the interaction of the HDI at various operating speed which gives another indication to the limitation of the numerical simulation.

In another similar approach to particle tracking in the HDD enclosure, Park *et al.* [16] investigates the characteristics of particle generation during the start/stop period using 5 different probes at different locations with the CNC as mentioned before. He investigated the effects of various sampling positions in the HDD enclosure and addressed the characteristics of generated particles at the different locations. These experiments enable the failure analysis to look into the visual damage location as viewed under the SEM or the AES. However, the limitation of these techniques still lies on the coupling of the visualization technique as mentioned in Section 1.3 to provide the path history of these particles before impact occurs. As experimental methods tend to be expensive and complex a cost effective way will be to use numerical simulation to predict the particle trajectories based on the computed flow fields. Numerical models can be used to experiment various scenarios in a cost effective manner to arrive at optimal designs for which experimentation could be developed

1.2.2 Outline of Computational Modeling of HDD Enclosure Flowfields

Among the first numerical investigation of the airflow in HDD enclosures was the work done by Chang *et al.* [17]. Using a finite difference method for solving the equations of fluid flow and incorporating the k - ϵ turbulence model, they showed good agreement between experimental and computational results with regard to the mean flow velocity and heat transfer characteristics in the disk driven enclosure.

Humphrey *et al.* [18] presented a three-dimensional numerical study of the unsteady flow. They computed toroidal vortices at the shroud acquire a time-varying sinuous shape in the circumferential direction. Suzuki *et al.* [19] numerically analyzed the effect of a radially inserted actuator arm and an “airlock” (which is similar obstruction to the flow). They computed the pressure distribution, shear stress distribution and disk torque coefficient from their computed flow fields. Iglesias *et al.* [20] performed two and three-dimensional calculations for different Reynolds numbers. Kazemi [21] conducted two and three-dimensional numerical calculation of the flow around a suspension-head unit and its vibration using a finite-element technique. Yuan *et al.* [22] also computed the effects of airflow pressure and shear forces on a rotating deformed disk in an enclosure with openings. The result was then used to develop a proper dynamical model for the investigation of the self-excited vibration modes of the disk. It was also observed that the magnitude of the vortex strength around the rotating disk increases as the Reynolds number increases.

Ali *et al.* [23] investigated the wall shape effect on the airflow characteristics in HDD enclosures.. The disk-to-wall gaps and curvatures of walls in a two-disks HDD model are varied to study the changes in the flow variation. In this study, results from an experimental work involving shrouded co-rotating disks configuration reported in Wu and Chen [24] shown in Figure 1.10 were used to verify CFD simulation. Figure 1.11 compares the computed variations of the normalised mean circumferential velocity as a function of the non-dimensional radial position, r^* with the experimental values obtained by Wu and Chen [24], Abrahamson et al. [4] (1989) and Tzeng et al. [25]

Ali *et al.* [26] also demonstrated the potential of CFD in predicting the effect of moving arm on the distribution and behavior of sub-micron particles within the HDD enclosure by coupling an aerosol model with the fluid dynamics model. It was found that the localized region in the vicinity of the HDI is likely to experience greater changes in airflow due to motion of the arm and suspension and these effects on the particles behaviors are well captured by the aerosol dynamics. Other computational studies in this HDD area include the works of Ng *et al.* [27], Shimizu *et al.* [28] who used large eddy simulation (LES) to study flow induced disk flutter and airflow induced vibrations of the HGA, Tsuda *et al.* [29] used Direct Numerical Simulation (DNS) for the simulation of HDD enclosure flows and Tatewaki *et al.* [30] reported LES results of airflows in realistic disk drives. Also recognizing that the airflow in a HDD is highly unsteady and random, most researchers have performed unsteady (time-marching) calculations, typically LES (or where resources permit, DNS). The converged airflow results are also used to simulate particles trajectories in the HDD enclosure. Computed results based on the Reynolds Averaged methods (which are useful in predicting mean flow fields and particles trajectories) have also been reported by Song *et al.* [31]. Goh *et al.* [32] predicted particle trajectories around the slider vicinity with a complicated model taking into account of the suspension details and slider geometries. They also observed that heat transfer into the flowfield as a result of prolonged operation from the motor-hub wall introduces thermophoretic forces on sub-micron particles. Most commercial software developed based on the Navier-Stokes (NS) Equation, which are valid almost everywhere inside the HDD enclosure. However, the airflow in the gap between the slider and disk, which has a thickness of the order of nanometer cannot be accurately described by the continuum model which breaks down at this scale. Figure 1.10 shows the configuration of the surface of a typical slider in perspective view.

The recession under this slider plays an important role in maintaining a constant fly-height during data transfer across the slider to the disk via the head located at the trailing edge of the slider. Traditionally, macroscopic hydrodynamic equations (e.g., Navier-Stokes, Reynolds) have been used to model slider air-bearing problems (HDI). In 1867, Maxwell discovered the “velocity slip” effect near a moving wall. The slip correction condition was introduced into Reynolds equations by Burgdorfer [33] in 1959

for Knudsen numbers lower than one ($Kn < 1$), where Kn is defined as the ratio of the molecular mean free path to the characteristic length of the flow. Huang and Boggy [34] use the compressible Reynolds equation with various slip corrections for rarefield flows, which was solved by a multi-grid control volume method for the simulation of arbitrarily shaped slider air-bearing with multiple recess levels. Their results were compared with the numerical solution of the direct simulation Monte Carlo (DSMC) method which, was first used for the slider air-bearing aerodynamics by Alexandra *et al.*[35]. Zhang and Boggy [36] had assessed the effects of lift on the motion of particles in the recessed regions of a slider (HDI). Figure 1.11 shows the coordinate system used in this analysis. As the particles in a recessed region are dilute in number, this study basically focused only on a single particle, neglecting the collision effects among particles.

The airflow in the recessed region is also regarded as laminar because the thickness of the air bearing is only a few micrometers in the typical sliders used in the industry currently. A simple slider with a flat surface is adopted in the calculation to simulate actual condition in the recessed region. For convenience, the perimeters of the slider are also assumed an “infinitely wide, no-rail slider” with some parameters fixed close to those of a recessed region. The solution of the dynamical equations of particle motion is done by first solving the Reynolds equation to obtain the pressure distribution P and then to calculate the velocity field V_g as a given condition. The pressure profile is obtained as shown in Figure 1.12.

The Reynolds equation is solved with the finite difference method. Figure 1.13 shows the effect of the relative velocity of a particle with the same density $\rho_p = 4000 \text{ kg/m}^3$ and different initial velocities of $U_{p0} = 0.4, 0.5, 0.6, 0.8$. It is seen that the particle with high (or low) velocity, goes up (or down) sharply when it enters the recessed region. The larger the magnitude of the relative velocity, the more sharply the particle moves up or down. It is also noted that a particle usually has a large magnitude of relative velocity when it just enters the recessed region. Then, it accelerated or decelerated by the action of the drag and moves gradually close to the velocity of the local airflow. Eventually, the magnitude of relative velocity gradually decreases towards zero. It is worth mentioning here that the initial velocity does contribute to the

trajectories of particles in the recessed region. An interesting study would be to understand the transient condition as the arm moves across the disk.

Using a force balance model, Tsai *et al.* [37] also found that the drag force on a deposited spherical particle, which is present inside the HDI is substantial to the detachment of the particle. He used a critical moment model to calculate the critical conditions for detaching a smooth spherical particle from a smooth solid surface. It was later found that the detachment of sub-micron particles from the disk surface by a slider air bearing is actually possible. With the increase in areal density thus the requirement of a lower flying height of the slider to achieve higher resolution is necessary. This lower fly-height is essential to maintain stability and better resolution of data transfer to the disk. A lower fly-height leads to the importance of analyzing particles in the nanometer scales as these particles will possible get caught in the HDI and causes damages to the read/write head. In reality, aerosol particles are present in a variety of sizes; i.e., the aerosol is polydisperse. Monodisperse aerosols are very rare in nature, and when they do appear, generally they do not last very long. It is also easily observed that at locations where there is a higher aerosol concentration, especially near the source zone, individual particles may coalesce to form larger chains made up of many particles. The process of coagulation may be brought about solely by the random motion and subsequently result in particle collision or collisions caused by external forces such as turbulence. As such, the aggregation or agglomeration of aerosol coagulation or inter-particles effects cannot be neglected.

The aerosol model uses the Eulerian frame of reference for presenting the particle concentration and size distribution have also been studied by many. Garrick *et al* [38] used a direct numerical simulation of a coagulating aerosol in a two-dimensional, incompressible, iso-thermal shear layer to study the nanoparticle coagulation. The evolution of the particle field is obtained by utilizing a nodal model to approximate the aerosol general dynamic equation (GDE). In another approach, Ali *et al.* [39] proposed an alternative method for predicting airflow in the HDD enclosure in an Eulerian frame of reference for representing particle concentration and size distribution. Computational results demonstrated that the Eulerian based model correlates well to the theoretical and experimental observations. There is also a general agreement between the result of the

Eulerian based model and those of the Lagrangian based technique used for comparison, indicating the suitability of the proposed model in predicting the airflow induced particle contamination in enclosures. In the macro flow outside the recessed region of the HDI, Bogy *et al.* [40] also investigated into the flow characteristics numerically. He observed that re-circulation starts to form near the edges of disks when the rotation speeds are large (small Ekman number) and the source strength is small (small Rossby number). Boundary layer equations thus cannot correctly describe the flow there.

Figure 1.10: Model used for the validation study: (a) cross-section of the shrouded co-rotating disks (Wu and Chen [24])

Figure 1.11: Radial Profile of the circumferential mean velocity [23]

Figure 1.12: 3D rail topography of slider with recesses on surface

Figure 1.13: Coordinate system

Figure 1.14: Pressure profile under a flat slider

Figure 1.15: Effects of the initial velocity on the path of particles: 1- $U_{p0} = 0.4$; 2- $U_{p0} = 0.5$; 3- $U_{p0} = 0.6$; and 4- $U_{p0} = 0.8$

1.3 Objective and scope of this research

The aim of this research is to understand the airflow characteristics inside a small form factor HDD enclosure subjected to a constant disk rotation and with the arm seeking across the disk from the hub (inner diameter-ID) to the outer diameter (OD) of the disk edge during the HDD read/write operation. The computed airflow patterns are then used to predict the trajectories of particles of various natures and sizes. Aluminum oxides, stainless steel and/or Poly-Oxymethylene (POM) are among the commonly found particles inside the HDD enclosure from drive's failure analysis.

The motion of the fluid in the HDD enclosure can be described, either by observing or predicting the trajectories of particles that are carried about with the flow which yields a Lagrangian description or by observing and predicting the fluid velocity at fixed point in space which yields an Eulerian or field representation of the flow.

This research seeks to analyze the accurate localize flow characteristics that must be obtained by solving the global flow field along with an almost exact geometrical representation of the space inside the HDD enclosure. The global airflow is set to spin by the disk rotation at low rotational speed.

The three-dimensional computational model consists accurate representation of the major components inside the HDD enclosure (Actuator arm, disk, magnet, latch etc.). The focus of this model is on the construction of the complex geometrical shape of the arm, providing detail accuracies towards the read/write head. A pseudo head-disk-interface (HDI) that is described by a $5\mu\text{m}$ air film is included in the domain. The length scale of the HDI used is the limit where continuum-based flow modeling is valid in this model. To further reduce the HDI spacing will reduce the Knudsen number to the transition continuum regime ($0.1 \geq \text{Kn}$) where continuum equations break down. Benzi and Damodaran [41,42] have recently investigated the airflow in the head-disk interface (HDI) of the hard disk drive (HDD) using parallel Direct Simulation Monte-Carlo model and presented on the different Knudsen numbers representation at the different regions.

Where most of the current research works deal with simplified HDD geometries, this work attempts to develop a complex geometrical mesh model to enable realistic prediction on the airflow and thus accurate particle trajectories. The co-existence of both a complex computational model of the HDD enclosure with the true geometry on the arm

assembly and thereby the HDD has significant contribution to the field of HDD research and development. The development of such a complex numerical model capable of predicting particles trajectories inside the HDD enclosure would facilitate a good tool for the design processes, failure detection of the HDD and also serving the platform for development, design optimization and parametric studies in a cost effective manner. Although significant progress has been made on the HDD, it must be noted little has been performed on a small form factor. Most of the earlier works focused on a 3.5-inch HDD. As the 1-inch HDD has only a short history since its production in 2000 by IBM, little research work has been done based on a complex model.

1.4 Layout of the thesis

The thesis is arranged in the following manner. Chapter 2 will illustrate the CAD modeling development process and presented a description to the method of meshing and trade-offs with a highly unstructured mesh. Much attention is focused on the modeling of the Head Stack Assembly (HSA), which includes the slider, suspension, lift-tab, gimbal assembly and the flexure. Chapter 3 will discuss the numerical model for the study of airflow patterns and particle trajectories in the HDD enclosure. Chapter 4 will present the computed results discussions pertaining to various issues such as airflow characteristics, particle trajectories predicted from the computed flow fields. Chapter 5 will conclude the study and propose recommendations for future work.

detailed region of the arm with the simple disk region are adopted. The latter is performed to control mesh gradient in the final volume mesh. Two significant advantages of using unstructured mesh are a reduced setup time then the structured setup and importantly the ability to incorporate solution-adaptive refinement of the mesh. By using solution-adaptive refinement, cells are added where they are needed in the mesh, thus enabling the features of the flow field to be better resolved.

2.1 Overview on modeling process of the HDD enclosure with artifact boundaries

Before going into the detail of the geometry of the HDD assembly, it is instructive to discuss the general assembly meshing process in more detail. Figure 2.1 shows the solution process for a typical CFD analysis. The three views (top, front, and side) and isometric drawings of the HDD enclosure obtained from the design engineers are translated into three-dimensional CAD model using *Rhinoceros 3D* [43]. Upon the completed constructing of all the surfaces, they are joined to form a sealed volume. The CAD model is then checked to ensure that all the surfaces are well connected without leakage or surface discontinuities within the volume. Once an “air-tight” model is recognized, the HDD assembly model is exported into *Gambit* [44], a commercial mesh generation tool via a *Standard for Exchange of Product Model* (STEP) format [45].

The use of automatic meshing scheme in *Gambit* allows the creation of the complete volume mesh in one step. However, in order to take full advantage of automation capabilities, pre-processing of the surfaces, and the geometric placement of nodal points must be improved. Nodal points of the right distribution are pre-assigned to the edges of surfaces so as to have better control of the surface mesh quality. Then, using these surface elements, the HDD model is packed with three-dimensional unstructured tetrahedral mesh. The completed domain is constantly checked and refined for three measures of quality as given by; mesh skewness, based on an ideal equilateral volume; smoothness, change in size should always be gradual; and/or cell aspect ratio, where the ratio of longest edge to shortest edge length should be close to unity. Usually, mesh refinement works are inevitable and it takes a few iterations before the domain is finally discretized for the implementation of the flow algorithm for computation.

The three-dimensional tetrahedral elements is then exported as mesh file into commercial solver, *Fluent* [46], where the flow field is computed. The various physical conditions defining the flow characteristics are defined within the flow solver. The initial and boundary conditions on each and every boundary within the HDD enclosure are also defined with respect to a moving wall or stagnant wall, heat source or non-heating wall. Convergence is tracked with a history plot showing all the calculated parameters. Usually, the set point that these flow parameters reaches convergence is pre-determined. The output data file is then exported into either a third party software, *EnSight* [47] or using existing post-processing capability inside *Fluent* for its data analysis.

Another alternative for acquiring the CAD model from interpreting the drawings is to performed modifications to the available mechanical CAD model from manufacturer. HDD assembly model can be imported directly into the meshing tool from the CAD system, however it usually appears in a form that is not directly meshable. This method would result in much more time spend performing geometry operations, both to make the HDD model meshable as well as to remove design details not relevant to the analysis being performed. The imported geometry not only contains numerous redundant details, but also contains numerous volumes and tends to be more complex in terms of surface topology. An example shown by Figure 2.2 is the wired view of the HDD mechanical model encompassing its full details. The motor-hub contains the internal structure of the various body components, like the female and male portion of screws and also capturing exact dimension of interference spacing. For the arm assembly, similar details are found at the pivot bearing describing the screw assembly. The flexures running along the suspensions are unnecessary details for flow calculations in the enclosure. The different layers of the filter are also redundant to the study of airflow inside the enclosure.

The amount of effort spent to account for these fine geometrical details into the mesh model do not provide much benefit toward improving the flow resolutions, as they do not attributes to change the overview of flow distribution significantly. Instead, large amount of time is spent in performing de-featuring or repairing. The primary approach was used for this modeling and the CAD airflow model is shown in Figure 1.2 of chapter 1.

Workflow diagram for CFD analysis

Figure 2.1: Workflow diagram illustrating the major steps in CFD setup.

Figure 2.2 - Geometric model of HDD directly imported from CAD system

2.2 HDD assembly model mesh generation

In this section, the unique features of the various components inside the HDD enclosure are presented and discussed with consideration to the final mesh. A layer of micron size mesh is assigned at the HDI, and controlled using defined grid boundaries and neighboring two-dimensional elements on the mesh smoothness and cell aspect ratio.

The arm assembly in the HDD is distinctly the most important component in the operation system as it carries the read/write head on the slider where magnetic signals are transfer across to the disk. To capture the flow around the HDI, it is inevitable that the modeling work including the elements that are packed around the vicinity must have good quality. Figure 2.3 shows a rendered view of the arm assembly used in the simulation. The entire assembly consists of four major parts, the suspension, arm, pivot bearing and the rear fin. The aim is to capture the outline of this assembly.

The three-dimensional view showing the HGA is shown by Figure 2.4. The slider mounted on the assembly is modeled as a rectangular block, which measures 1.25mm in length by 1.01 mm in width and has a 0.33mm thickness. The slider is swaged to the flexure and pivoted at the dimple as shown in the picture. During read/write operation, the slider induced by the wind flow and mechanical vibration, emanated in pitch and roll about the dimple as indicated by the arrows.

Pitching here refers to the rotation of slider about the X-axis while rolling refers to the rotation of slider about its central Y-axis. Pitching and rolling are important characteristics of the slider as in the event of shock, provides the slider with self-stabilization. Figure 2.4 shows a limiter that is rigidly attached to the slider. It acts as a latch providing a breaking effect in the event of a large pull off of the slider from the disk during unloading, which otherwise may cause plastic deformation to the thin flexure material. This breaking action also prevents large displacement due to vibration thereby also possibly the disk.

As the arm consists of many steps and turns and is made up of an assembly of several layers of components, it will be difficult to use quadrilateral elements on its surfaces. Figure 2.5 shows the triangular elements attached on the HGA. The qualities of the surface elements were inspected and showed good results for skewness, cell aspect ratio and smoothness. The slider is omitted in this drawing for clarity of presentation.

The surface meshes on the HGA are necessary to maintain a small size interval especially towards the lift tab keeping in mind that a layer of 5 μ m tetrahedral volume mesh is required to fill the HDI. This is to avoid a large transition of size growth from the HDI as mentioned earlier that surface elements are used as growth source from where volume meshes are generated in that region.

To facilitate better quality volume mesh around the suspension, quadrilateral elements are used to replace triangular elements on uniquely long surfaces with extremely high aspect ratio. An example is the side rail of both the flexure and suspension along the longitudinal axis of arm, of which surfaces are extremely high in terms of aspect ratio.

Figure 2.6 shows the top/bottom sliders, meshed with triangular elements and as they approach the disk side, the size interval defining the edges reduce until they match with an edge length equivalent to 5 μ m gap. Ideally, this gap should be challenged to the finest that the N-S solver could handle. The air bearing gap due to the windage of the spinning disk in the HDI is close to 5nm, which will not be discussed here as the Khudsen number goes above unity in that region therefore limiting the continuum equation from predict accurately on the flow physics.

Figure 2.7(a) shows the side views to the dimple-slider in the HGA and the Figure 2.7(b) shows the respective surface mesh. It is important to model correctly with a good description by the grid and then fine mesh. Due to the resistance of the slider to the spinning disk flow, it would be predictable large changes in flow parameters in this region. A fine resolution of elements in this region improves the prediction of flow features when going closer to the HDI.

Figure 2.8 shows the grid defining the flexure. Slider will be attached to the rectangular area from the bottom. The flexure has a thickness of 0.02mm and therefore surface elements adjacent to the edges must be limited to below 0.1 size interval equivalent to 0.03mm in edge length. The side rails are meshed with quadrilateral elements to maintain good skewness and lesser quantity are required. The structure is bent upward and resembling a T-Shape limiter.

Figure 2.9 shows an intricate suspension with its side folded into a wedge and also consisting openings at the center structure that attributed weight reduction in the final

assembly. One problem with the construction of structure with numerous intricate surfaces is these non-linearity surfaces usually constitute large tolerance error due to interpolation errors at its edges and poses difficulties in connectivity. Remember that discrete surfaces need to be joined to form a close volume. Figure 2.10 shows a typical case of geometrical tolerance error, which requires the reconstruction of surface or even volume. The highlighted edges indicate surfaces discontinuities. In such cases, it is imperative to recreate the adjacent surfaces. An easy way to force merge these edges can be performed using the modeling tool whichever way is achievable. In the latter approach, force merging of surfaces at their edges may not succeed if numerous surfaces errors exist. The summation of dimension tolerance in a single volume would be considered below limit.

In this study, a simple HDD enclosure model discarding all intricate contours is also developed for parametrical studies. The baseline to such CAD modeling falls to the topology shaping. The simple model will be used to validate against the complex model results and discussed the effectiveness of model based on computational flow results in Chapter 4. The major changes from the complex model lies greatly by the arm shape. Figure 2.11 shows the suspension of the simple arm model at the Inner Disk (ID) location. The control of the surface mesh distribution on the motor hub wall is important to ensue the smooth generation of three-dimensional hexahedral elements at the slider vicinity. Where fine mesh control is possible on the hub wall nearer to the suspension, it should be refined. This helps the mesh generator to recognize that smaller volume mesh should be assigned in that region. The slider remained the same dimension to the complex model and HDI gap is unchanged. Figure 2.12 shows the mesh distribution from the top view at the arm region. Higher density of small size elements are packed at the suspension and gradually expanded from the HDI. Figure 2.13 shows the top view of the mesh elements on plane 1. In this plane, the finer mesh is observed at the arm, latch, the thin space between the magnet and arm rear fin where surface elements are small due to fine gap.

Figure 2.3: Rendered view of arm assembly from modeling software

Figure 2.4 – Perspective view of the Head Gimbal Assembly (HGA)

Figure 2.5 – Unstructured triangular surface mesh on the HGA

Figure 2.6 – Surface mesh on slider with size function on mesh growth rate

Figure 2.7 – Dimple & slider (a) CAD model (b) surface mesh

Figure 2.8 – Flexure and limiter

Figure 2.9 – Lofting the arm suspension

Figure 2.10 – Unmerged edge on the arm suspension

Figure 2.11: Surface mesh on rotor-hub wall with gradual refinement towards disk (Disk is omitted for clarity of presentation).

Figure 2.12: Close up at suspension vicinity. Mesh is refined in the vicinity of slider

Figure 2.13: Unstructured grid displayed on plane 1 between top disks and slider

Figure 2.14(a) shows the perspective view of the load/unload (L/UL) ramp. The slope is inclined at 16° for top/bottom from the disk plane.

Figure 2.14(b) shows the assembled of the L/UL ramp with the disk and arm. The ramp has significant impact on the flow distribution as part of its assembly cuts into the disk region. It poses a resistance to the rotating flow and was identified as a particle generator spot due to possibly abrasion generated particles from long-term contact with the lift tab during L/UL operation. The space between the ramp edge and the disk surface for both top/bottom sides is measured at 0.195mm. The obstruction of the ramp at the high angular velocity region in the outer disk anticipates localized turbulence and possibly induced modulation on the suspension.

Figure 2.15 shows the ramp grid meshed with triangular surface elements. A size function is attached to the edges along the ramp slope as indicated in the figure. Nodes interval of 0.1 was assigned to the edges furthest into the disk. Moving away from the disk, a interval growth factor is set to increase the interval distance between nodes by a factor of 1.1 for each increment and limited by interval value of 0.5. Surface meshes are then activated to fill these surfaces with a constant interval size of 0.5 except where predefined by the node intervals. The elements in the small gaps under the slope should be refined to avoid high skewed element during the volume mesh generation.

Conventional HDD is not totally sealed from the ambient air, but they are definitely separated from it. In a actual HDD operating condition, a breather filter of certain permeability would bridge the connectivity between the ambient air to the enclosure airflow for two reasons, a: prevent large humidity changes thus water droplet forming into drives and b: filtrating organic vapors into the enclosure. The filter is usually packed with carbon compound contain in a permeable membrane, and sealed by an adhesive on the top cover breather hole. Carbon deals with the removal of unwanted vapors from the airflows and aids in the purification process of the flow leaving the absorptive cell. Piscitelle *et. al.* [48] has developed a model based on equations derived by Amundson and Bohart [49] to describe flow in carbon filter beds. This working model accurately predicts both the breakthrough time and the shape of the breakthrough curve. Nir *et al.* [50] has also studied the processes of dynamic absorption of toxic vapors on activated carbon filters using fixed airflow rates and pulsating flow rate.

Figure 2.16 shows the three-dimensional views of the breather filter from two different angles. The center portion measures 0.4mm in thickness and is encapsulated by a layer of thin adhesive. The boundary condition of this filter is set as a rigid wall and non permeable. Breather filter has also the function to remove unwanted vapors and is also equally important to provide particle removal capability in the HDD enclosure.

Particles inherited from assembly process could be caused from numerous sources. These residual particles can exist in various forms and locations in the HDD enclosure. Residual particles are swept into circulation carried by the induced air stream from the spinning disk. The re-circulation filter appears in a pouch form and is meant for fast removal of vapor, particle contaminant and also protects against spike contamination. Figure 2.17 shows also the modeling of the re-circulation filter, as well as the shape of the holder. The boundary wall of this filter is set as a porous jump boundary. Porous jump conditions are used to model a thin "membrane" that has known velocity (pressure-drop) characteristics. It is essentially a one-dimension simplification of the porous media model available for cell zones. Examples of uses for the porous jump condition include modeling pressure drops through screens and filters, and modeling radiators when heat transfer is not a concern to the problem. Porous jump boundary condition is a simpler model because it is more robust and yields better convergence. The thin porous medium has a finite thickness over which the pressure change is defined as a combination of Darcy's Law and an additional inertial loss term:

$$\Delta p = - \left(\frac{\mu}{\alpha} v + C_2 \frac{1}{2} \rho v^2 \right) \Delta m \quad 2.1$$

where μ is the laminar fluid viscosity, α is the permeability of the medium, C_2 is the pressure-jump coefficient, v is the velocity normal to the porous face, and Δm is the thickness of the medium. The values for appropriate values for α and C_2 are determined as $5e-07 \text{ m}^2$ and 0. The porous media thickness is 0.001mm. The two-dimensional boundary face that is created within a three-dimensional space can be performed with the split operation in Gambit during the assignment of boundary condition.

Figure 2.18 illustrates a filter media with different layers of materials that is used for filtration optimization. Filter combined with electrostatic media provides effective particle clean-up performance along with its activated carbon to absorb harmful chemical vapors and protect critical drive components. Harmful chemical vapors may also be a result from out-gassing of component material or vaporization caused by spindle motor oil in long operating hours.

Figure 2.19 (a)-(b) shows the exploded view of the latch and the pivot pin. The latch functions as the lock to arm assembly when the HDD is in the stop mode. It prevents the slider on the suspension arm from releasing down the ramp and come in contact with the disk thus causing scratches to both surfaces during the next start up. The latch design can be used to study the effluent behavior of air from under the breather filter and a potential particle accumulation and release point.

Figure 2.20(a)-(b) show a close replication for the top cover shape between a true model and the modified simple model. In Figure 2.20(b), the magnet, along with the latch are mounted onto the top cover as a single unit. The openings to the top cover were sealed for the final model, however retaining the surface topology. Similarly, Figure 2.21(a)-(b) shows a close resemblance of the two models of the motor base between the actual and the constructed component. The details on the screws inside the motor base and exterior shapes are not accounted in the final model.

Figure 2.22 presents the overview of the mechanical assemblies of HDD with an opened top cover. All components are laid out in their respective positions and unionized to each other to form a single volume. The assembled HDD model is then inspected for discontinuities at connecting edges to ensure an “air tight” volume. The successful merging would create two volumes, one domain represents the mechanical model and the inner volume would be representation of the enclosure airflow domain. The mechanical domain is deleted. Figure 2.23 (a)-(b) show the transformation of the structural domain to the airflow domain. The resultant geometry is still not ready for meshing until all edges are cleaned.

When two vertices from each volume coincide during Boolean operations (union or split), it is termed coincident control point. This is illustrated with an example as shown by Figure 2.24, where the vertex at the tip of the cone and edge of a block meet at

the same Cartesian point. The coinciding of the same vertices point resulted in a singularity point that forbids successful Booleans operation. The affected faces have to be re-constructed with different control points on the affected surfaces. The repairing job can be labor intensive especially when the geometry is complex.

Figure 2.25 shows the complete mesh of the HDD enclosure model. The initial volume mesh domain usually contains a lot of geometrical errors and will not converged due to the low quality mesh. Constant checks on the volume mesh qualities based on high skewed angle, poor cell aspect ratio and cell gradient smoothness are essential. Dirty geometries are usually associating with surface cracks, overlaps or invalid manifolds before grid generation can take place. Bad surface refers to discontinuities at edges thereby unable to form a close surface during Boolean operation.

Figure 2.14 (a) Perspective view of load/unload ramp (b) Ramp assembly in the HDD

Figure 2.15: Surface mesh on ramp

Total thickness ~ 0.4 mm
Air filter dimension was
estimated from physical drive

Figure 2.16: Breather filter modeling

Figure 2.17: Re-circulation filter holder

Figure 2.18:Exploded layers of re-circulation filter [51]

Figure 2.19: Rendered views in different angles (a) constructed in Rhino [43] (b) extracted from Seagate Isometric drawing

Figure 2.20: Rendered view of top cover (a) extracted from Seagate Isometric drawing (b) constructed in Rhino [43]

Figure 2.21: Rendered view of motor-base (a) extracted from Seagate Isometric drawing (b) constructed in Rhino [43]

Figure 2.22: The interior wall shape of HDD enclosure generated by Rhino [43]

Figure 2.23: CAD models of (a) HDD mechanical structure (b) Fluid region inside HDD enclosure

Figure 2.24: Interception of a cone and block volume with coincident control point

Figure 2.25: Surface meshes on complex HDD enclosure walls

2.3 Software interface errors

The transferring of CAD geometries between software introduces geometrical errors due to dimensional tolerance and mismatch of surface caused by a shift of control points. Figure 2.26(a) shows the top view of the dimple point on the suspension in CAD modeling software, Rhinoceros. The same grid was exported into Gambit for meshing purpose. It was observed the changes on certain surfaces contour due to the translating of control points. The exported surface as shown by Figure 2.26(b) shows interpolation error due to point shift. These errors change the original shapes, making it harder to mesh due acute angles between edges. The error surfaces are deleted and rebuilt from various sources usually taken from adjacent surface edges.

Bad surfaces with sharp edges often results in meshing difficulties and is highly recommended to reconstruct instead of utilizing existing functions in modeling tools for its repair which normally end up substantial time in trials & errors and bad results. A value of >0.85 skewed value is considered upper limit when performing quality check on the two-dimensional elements. As three-dimensional elements qualities are highly dependent on surface mesh, poor surface meshes percentile would deteriorate the volume mesh further and results in convergence difficulties.

Solutions to the mesh skew problem have been explored [52]. While a workable solution has not yet been completed, it will likely include adding virtual vertices, new interval assignment constraints, and/or modifying the interval assignment optimization process. Solving the mesh skew problem will improve both on the ease of convergence and overall mesh quality, therefore leads to shorter time.

Figure 2.27(a) shows an example of a bad surface that has a highly acute angle from close edges. High skewed elements can be improved by assigning more nodes or adjustment of nodes distribution on edges, alternatively to rebuild the surface. Figure 2.27(b) shows the skewness quality check in color codes. Blue shows good skewed angles and red indicates the extreme bad elements. Increasing nodes to form equilateral triangles may improves the qualities however, increases in computational requirements for more elements generated. Figure 2.28 shows the reshaping of the e-block to avoid sharp angles between two surfaces.

Figure 2.26: Wire frame view of dimple on suspension in (a) *Rhinoceros* (b) *Gambit*

Figure 2.27: Suspension in (a) Wire frame view (b) mesh quality inspection mode on skewness check

Figure 2.28: Modified edge on arm suspension to eliminate acute mesh angle

2.4 Automatic volume mesh scheme

To assess the effect on different meshing approaches in discretizing complex domains, both single and split volumes of the HDD have been considered.

The automatic meshing with “Tetrahedral and Hybrid” scheme are chosen to achieve uniformity in volume mesh type. This scheme specifies that the volume meshes are composed of tetrahedral, hexahedral, pyramidal, and wedge elements where appropriate within the geometry. Gambit allows the specification to any volume for a meshing operation; however, the shape and topological characteristics of the volume, as well as the vertex types associated with its faces, determine the type(s) of mesh scheme that can be applied to the volume.

To specify the meshing scheme, three parameters elements, type and smoother for map meshes must be specified. The elements parameter defines the shape of the elements that are used to mesh the volume. The Type parameter defines the meshing algorithm and, therefore, the overall pattern of mesh elements in the volume. The Smoother specification determines the type of smoothing algorithm (if any) used to smooth a mapped mesh during the meshing operation.

Figure 2.29 (a)-(b) show the discretization of the HDD enclosure for a single grid volume treatment and split volume grids as mentioned earlier. GAMBIT attempts to create a mesh that consists primarily of tetrahedral mesh elements but which can also contain hexahedral, pyramidal, and wedge elements where appropriate. Hexahedral, pyramidal, and wedge elements are typically created in regions that are adjacent to pre-meshed faces and/or affected by pre-existing boundary layers.

As an example of the T-Grid meshing scheme, consider the cubic volume shown in Figure 2.31(a). The faces of the cube are unmeshed and the volume does not serve as an attachment entity for any boundary layer. If the volume mesh is performed by means of the T-Grid scheme, GAMBIT creates a mesh composed solely of tetrahedral elements such as those shown in Figure 2.31(b).

In section 2.2, it was mentioned that the side rail of the suspension arm were pre-meshed using quadrilateral face meshes. The use of T-Grid scheme (by default) creates a layer of pyramidal mesh elements adjacent to the pre-meshed face. Figure 2.32 shows a

layer of pyramidal elements adjacent to the pre-meshed suspension edge created with the T-Grid scheme. Tetrahedral elements are then produced throughout the rest of the enclosure where possible.

The difficulties to achieve a single volume mesh lies in the control of mesh smoothness in the internal transition from fine meshes zone to coarse mesh, between extremities. A single volume model has lesser surfaces to be used for pre-mesh adjustment to the final mesh. Also, as no split of volume is necessary, the final mesh retain the shapes of all the surfaces thus achieving a better quality 3D mesh

The decomposition-based approach to tetrahedral mesh generation results in a large number of volumes in the final model. In this case, we strictly stick to four volumes due to complexity of the problem. Typically these volumes are meshed with a variety of different meshing algorithms, depending on their geometry and topology. There are benefits in split volumes as compared to a single volume in terms of mesh qualities control, smoothness, skewness, etc. By splitting volume, it uses lesser memory each time a partial volume is meshed and therefore, higher density meshes is possible to achieve. There are two methods in splitting a volume. One method is to split a volume into half with their respective boundary surface belonging to each other. The other method is to half an existing volume and sharing the interface boundary. In this way, the two volumes are recognized as one entity in Fluent. The interface is invisible to the solver. Each of which is presumable easier to mesh than the original part.

Figure 2.33 shows the velocity contours on a selected plane in the xy axis. The meshes at the boundaries of the split surfaces are refined to eliminate kinks and achieve flow continuity during the presentation on the computed flow variables. From the contour plot, no kinks are observed and the flow is not discontinuous at the boundaries.

(a)

(a)

Figure 2.29 – Volume mesh built from (a) a single HDD enclosure CAD geometry (b) split or multiple CAD volumes

Figure 2.30 – Splitting from single CAD volume

Figure 2.31: Simple T-Grid meshing scheme (a) Unmeshed cube (b) Simple T-grid mesh

Figure 2.32: Pyramidal elements adjacent to the pre-meshed suspension edge

Figure 2.33: velocity contour of split volume: Split boundaries are non-visible.

2.5 Mesh Generation at HDI

The airflow in the HDI has been extensively research in the last decade. Huang *et. al* [53] used the direct simulation Monte carlo (DSMC) method to solve the three-dimensional nano-scale gas film lubrication problem between a gas bearing slider and a rotating disk. Fukui and Kaneko [54] developed a more sophisticated slip correction for Reynolds equation based on the Boltzmann equation. In this work, the author attempts to incorporate a generic micro-air film under the block slider as a parametric study to the flow around the slider. Although it is not real, the presence of this thin layer contributes to changes of pressure and velocity field in the slider vicinity, enabling study of geometrical effect on particle trajectories. To enable meshing in the HDI, a layer of extremely fine meshes are packed into the space and grow gradually away from the HDI region.

The slider surface parallel and closest to disk is used as the boundary for sizing growth. Figure 2.28(a) shows the split top disk surface demarcated with the perimeter of the slider. The split edge on the disk is used as a medium to control nodes distribution and also the source point for element size growth. The intent of this splitting is to enable nodes assigning on the edge of the boundary on face 1 as evenly spaced to the slider's surface nodes to achieve a good skewed distribution of element in the HDI. Figure 2.28(a)-(b) shows the side of HDI and the possible connection of mesh via the distributed nodes.

Figure 2.28(a) shows an equal interval of node distance between the disk and the slider. HDI in this model is approximated 5microns. However, nodes intervals exceeding the HDI gap and/or varies between the 2 surfaces would result in highly skewed elements as highlighted by the red cells in Figure 2.28(b). To ensure the smooth gradient growth of the volume mesh from the HDI, size functions are assigned to surfaces whenever possible. Figure 2.28(b) shows the distribution of triangular elements on the disk. Elements of 0.005 size intervals grow from the boundaries of face 1 with a growth ratio of 1.08 to reach the limit of 0.5. A high density of elements resulted from the growth function gathers at the slider vicinity in the XY plane. Similarly in the Z direction, size functions are attached to the surfaces on the top/bottom sliders. Mesh elements increase from 0.005 with 0.2 with an increment ratio of 1.1 as shown by Figure

2.20. The disk is omitted in this figure for clarity of presentation. The edge near the disk has the same node interval of 0.005 with the disk.

The different volumes meshes are connected by creating non-conformal nodes interface between the shared faces. The split is avoided where continuity of flow is important. Splitting a volume is effective where intensive meshing on global geometry is avoided for only minor localized changes. Therefore, only certain part of the split geometry is necessary to be re-meshed. However, single volume mesh domain is inferior in term of re-meshing the entire volume with respect to changes in geometry where all other meshes would be removed and set to default.

Using the splitting volume approach, iterative designs that require many shape revisions to small regions can be repeated easily. Tremendous amount of time saved can be acquired for design process. Figure 2.30 shows the exploded view of the split mesh volume. The red dotted circles shows the sharp corners on surfaces that would result in high skewed surface element created during meshing. Also, it takes longer time to select appropriate boundaries for split and resultant volume may not be meshable if bad surfaces with high skew angle as a result of the operation.

The area of interest in this problem lies in the vicinity of the slider and therefore, an exact arm geometry is adopted with the mesh surrounding the slider became critical to obtain a realistic prediction.

The quality of mesh in this modeling plays a significant role in the accuracy and stability of the numerical computation. The attributes associated with mesh quality are node point distribution, smoothness, and skewness.

The degree to which the salient features of the flow, such as shear layers, separated regions, boundary layers, and mixing zones, are resolved, depends on the density and distribution of nodes in the mesh. Poor resolution in critical regions (slider vicinity, etc) can dramatically alter the flow characteristics. The prediction of separation due to an adverse pressure gradient depends heavily on the resolution of the boundary layer up stream of the point of separation.

Resolution of the boundary wall layer (i.e., meshing spacing near walls) also plays a significant role in the accuracy of the computed wall shear stress and heat transfer

coefficient. This is particularly true in this study where laminar flow will be assumed throughout the calculation. The grid adjacent to the critical walls should obey

$$y_p \sqrt{\frac{u_\infty}{\nu x}} \leq 1 \quad (2.1)$$

where y_p = distance to the wall from the adjacent cells centroid

u_∞ = free stream velocity

ν = kinematic viscosity of the fluid

x = distance along the wall from the starting point of the boundary layer.

Smoothness of the grid distribution depends on the gradient of cell volume between adjacent cells. Rapid changes in cell volume translate into larger truncation errors. Truncation error refers to the difference between the partial derivatives in the governing equations and their discrete approximations.

The importing of CAD model into Gambit by a various upon importing the geometry is the cleaning of dirty edges. Dirty edge in geometry refers to the extension of faces that residual from either a Boolean operation or change of file format and appears as a line with almost zero surface area. One can imagine a line drawn from a surface as shown by Figure 2.11, extended outside the surface area with hardly any measurable area. Dirty edges impede the generation of surface meshes & therefore prevent volume mesh.

These dirty geometries confuse the program during volume meshing and forms highly skewed 2D elements, which produces bad quality mesh and/or fail to initialize the volume mesh at all. Dirty edges are removed by activating the clean function in Gambit and/or irreparable circumstances, requires reconstruction of surfaces in Rhinoceros before re-exporting again into Gambit for verification.

Figure 2.12(a) shows the imported HDD geometry in Gambit. Usually, vertices resulted from Boolean operations in the geometry construction process will resurface from the importing process. Figure 2.12(b) presents a close up view on the lift tab. The vertices are manually removed by merging operations. Figure 2.13 (a)-(d) show the sequences of vertices removal process before creating surface meshes.

Figure 2.40(a) shows the default quadrilateral surface mesh generated with on the lift tab with unspecified mesh type and size. The face meshes colored by the yellow edge shows 2 quadrilateral face meshes were utilized to represent the curl contour of the lift tab. This would be undesirable as the accuracy of the region near the arm can only be achieved with more cells.

Gambit automatically evaluates the face with respect to its shape for the selected faces, topological characteristics, and vertex types and sets the Scheme to reflect a recommended face mesh scheme. As such, the resultant quadrilateral face mesh as shown by Figure 2.13(a) reflects the recommended scheme for the most recently specified face.

However, such an operation would treat every face, edge with equal intervals and eventually produce grids that are of poor quality. Figure 2.13(b) shows that the vertices along the edges been removed by a merged operation and thus minimizing the confusion for element growing out of these edge intervals which sometimes produces high aspect ratio elements. To set the face element type, the node pattern associated with each of the face element shapes must be specified. There are two face element shapes available in Gambit; (a) Quadrilateral and (b) Triangular. Each face element shape is associated with three different node patterns. The individual node pattern is then characterized by the number of nodes within. Figure 2.14(a)-(b) show the node patterns associated with the quadrilateral and triangular face element types, respectively. However, in this study, variation of nodes is adopted for each face due to the high complexity of the faces to volume growth. Figure 2.13(c) illustrated a 0.05 interval displacement of nodes assigned to all the highlighted edges in red.

In the face mesh option, it is essential to enforce suitable meshing scheme coupled with the type option to produce reasonable face representation. The final face meshes are shown in Figure 2.16(d). The “Pave” type-option meshing scheme with triangular elements creates an unstructured grid of mesh element.

A plane taken across the top HDI is shown by Figure 2.21. The grid shape maintains uniform to the surface mesh at 5 microns above disk. Outside the disk

boundary, mesh elements takes the size of the enclosure wall surface mesh. A large jump in mesh gradient might occurs from the disk to the surrounding region if the mesh at the walls nearest to the disk perimeter are not refined to prevent the gradient. In this case, the other walls nearer the disk perimeter have nodes limited to 0.7 intervals. However with the aforementioned, the grid quality can be further improved by locally densifying the mesh where high resolution of flow is required.

Figure 2.34: Nodes distribution in the Head Disk Interface (HDI) based on (a) evenly distribution nodes (b) Unevenly distributed nodes

Figure 2.35: (a) Top view of disk split with slider area (b) Mesh distribution on disk surface with smooth cells growth from mesh perimeter

Figure 2.36 (a) Surface mesh on arm (b) Size function attached on slider surface

Figure 2.37 – Schematic diagram illustrating a dirty Edge

Figure 2.38: (a) Real HDD volume generated with *Gambit* [44] (b) Close-up view on lift tab in *Gambit* [44].

Figure 2.39: Nodes distribution on lift tab with (a) Coarse face meshes from automated meshing scheme (b) vertices removal from edge (c) Nodes of fine intervals assigned to edges (d) Good quality unstructured triangular surface mesh

Figure 2.40: Types of face elements with (a) Quadrilateral (b) Triangular

Figure 2.41: Section of volume mesh across plane 1 (Between top disk surface and slider): The volume mesh size is limited to 0.7 interval to enable cell smoothness.

2.6 Mesh Adaption

The solution-adaptive mesh refinement feature of *Fluent* allows the refinement and/or coarsening of grid based on geometric and numerical solution data. The refinement to the grid near one or more boundary zones is utilized because important fluid interactions often occur in these regions. When adaption is used properly, the resulting mesh is optimal for the flow solution because the solution is used to determine where more cells need to be added. Thus, computational resources are not wasted by the inclusion of unnecessary cells, as it occurs in the structured grid approach. Also, the effect of mesh refinement on the solution can be studied without completely regenerating the mesh. The hanging nodes method is adopted and it is characterized by nodes on edges and faces that are not vertices of all the cells sharing those edge or faces as shown by Figure 2.41.

As explained with Equation (1), the grid size nearest to the boundary walls must not exceed the given calculated distance in order to capture the characteristic of the near wall accurately. Using the boundary adaption function, the meshes near the wall enclosure were adapted based on the normal cell layers. Figures 2.44(a)-(b) shows the refinement of grid near boundary wall with a slice of the 2D grid taken along the arm length from mid width. Figure 2.44(a) shows the distribution of coarse triangular grids

located at the enclosing wall vicinity. Figure 2.44(b) shows the improvement of the adapted cells with more refined elements next to the enclosure wall. Although it will be ideal if more layers are included in the refinement process, the grid resolution is usually constraint by the CPU time and computer resource limitations (i.e., memory and disk space). As such, it is important to determine the locations of important flow features in advance through experimental data or theoretical intuition to minimize the memory usage for adaption.

The flow at the vicinity of slider is important in this study. A high percentage of HDD failures are caused by flow-induced particles at the slider region. The arm modulation has the highest vibration frequency at the distal tip of the arm, where the slider experienced skip track errors and intermittent breakage of read/write signals. This is exacerbated when the arm approaches the OD where the lift tab is approximately only 2 mm distance away from the ramp slope. To augment the realism of the flow around the slider, the mesh element at the region is critical to be refined to capture a realistic salient feature. The choice of adaption based on velocity contour is considered. Further information on choice of adaption method is discussed in the result chapter.

A total of 510903 cells are marked for adaption as shown by Figure 2.43. The individual cells are marked for refinement based on the adaption function, which is created from the initial computed solution data. The cell is then refined or considered for coarsening based on these adaption marks. The primary advantages of this modularized approach are the abilities to create sophisticated adaption functions and to experiment with various adaption functions without modifying the existing mesh.

Tetrahedral with hybrid mesh generation scheme is a robust method to mesh complex volume. The pre-mesh process on surfaces enhances the final three-dimensional mesh qualities. Also, the trend in modeling and simulation towards more complex assembly geometry, with multiple parts is observed. These trend points toward an increased need for assembly meshing.

With the good control of nodes intervals on edges, good surface mesh can be observed. This leads to a good three-dimensional mesh being developed based on an all-tetrahedral/hybrid meshing scheme in Gambit.

Figure 2.42: Example of a hanging node

Figure 2.43: Grid distribution taken along the parallel axis of the arm at the mid width (a) before adaption (b) after adaption

Figure 2.44: 510903 marked cells for adaption

Figure 2.45: Grids (a) Before adaption (b) After adaption, clustering of cells at high velocity region

CHAPTER 3

COMPUTATIONAL MODELLING OF FLOW FIELD CHARACTERISTICS AND PARTICLE TRANSPORT IN HARD DISK DRIVE ENCLOSURES

In this chapter, the numerical schemes, which solve the flow field in an 1-inch HDD enclosure are briefly outlined. The pressure and velocity fields in the DD enclosure for the various static arm positions are acquired by solving the Navier-Stokes (NS) equation. Approaches to solve these algebraic equations are explained. The temporal discretization scheme for a scalar transport equation is also used to discretize the momentum equations. The discretization of the momentum and continuity equation with a pressure-based solver is addressed.

Based on the finite volume-based method (FVM), the computational solver numerically discretize the governing equations into algebraic equations, which are then solved by high performance computing systems using efficient flow algorithms.

Particle transport in the HDD enclosure is described by using particle-transport models. The two approaches for the numerical calculation of particles trajectories with the Lagrangian and Eulerian reference frame of tracking are introduced. The boundary conditions for the discrete phase during contact with boundary surfaces are also discussed. The discrete phase boundary condition is important to the fate of the trajectory at specific boundary wall contact and the particle contact condition can be defined separately for each zone in the model.

3.1 Flow Models

The flow field inside the HDD is solved by the unsteady incompressible Navier-Stokes equations, which consist of the continuity, mass momentum and energy equations. The incompressible Navier-Stokes (NS) equations are an adequate representation of the flow within the HDD enclosure as the Knudsen number and Mach number are lesser than 0.01 and 0.1 respectively. Both the compressible & incompressible Navier-Stokes equations are presented.

The equation for conservation of mass, or continuity equation in general form for compressible flows is written as shown by Equation 3.1(a).

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \quad (3.1a)$$

where ρ is density, $\vec{v}$ is velocity vector and t is time in seconds. The right hand side of the equation 3.1a is set to zero, assuming there are no additional source into the HDD enclosure domain from any user-defined sources. The incompressible form of the continuity equation where ρ is set constant is defined as,

$$\nabla \cdot (\rho \vec{v}) = 0 \quad (3.1b)$$

Conservation of momentum in an inertia (non-accelerating) reference frame for the compressible flows as introduced by Batchelor [55] is written as follows;

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \quad (3.2a)$$

where p is the static pressure, $\bar{\bar{\tau}}$ is the stress tensor where $\bar{\bar{\tau}} = \mu \left[(\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \nabla \cdot \vec{v} I \right]$, and $\rho \vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively. $\vec{F}$ also contains other model-dependent source terms such as porous-media and user-defined sources. In the $\bar{\bar{\tau}}$ term, μ is the molecular viscosity, I is the unit tensor, and the second term on the right hand side is the effect of volume dilation.

The incompressible form of the equation becomes;

$$\rho \left(\frac{\partial \vec{v}}{\partial t} + \nabla \cdot (\vec{v} \vec{v}) \right) = -\nabla p + \nabla \cdot (\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \quad (3.2b)$$

The energy equation for compressible flow is solved in the following form:

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{v}(\rho E + p)) = \nabla \cdot (k \nabla T - \sum h_j \vec{J}_j + (\bar{\bar{\tau}} \cdot \vec{v})) + S_h \quad (3.3a)$$

where k is the thermal conductivity, and $\vec{J}_j$ is the diffusion flux of species j . The first three terms on the right-hand side of Equation 3.1(d) represent energy transfer due to conduction species diffusion, and viscous dissipation, respectively. S_h includes the heat due to any other defined volumetric heat sources.

The total system energy, E in equation 3.3a is defined as

$$E = h - \frac{P}{\rho} + \frac{v^2}{2} \quad (3.3b)$$

For incompressible flow, the enthalpy h is defined as

$$h = \sum_j Y_j h_j + \frac{P}{\rho} \quad (3.3b)$$

where Y_j is the mass fraction of species j and

$$h_j = \int_{T_{ref}}^T c_{p,j} dT \quad (3.3c)$$

where T_{ref} is 298.15K.

The form of the generic transport equation for an arbitrary scalar ϕ_k for a single-phase flow is written as;

$$\frac{\partial \rho \phi_k}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i \phi_k - \Gamma_k \int \frac{\partial \phi_k}{\partial x_i}) = S_{\phi_k}, k = 1, \dots, N \quad (3.4)$$

where Γ_k and S_{ϕ_k} are the diffusion coefficient and source term for each of the N scalar equations. The diffusion coefficient Γ_k is defined as a tensor in the case of anisotropic diffusivity. The diffusion term is thus $\nabla \cdot (\Gamma_k \cdot \nabla \phi_k)$. For isotropic diffusivity, Γ_k could be written as $\Gamma_k I$, where I is the identity matrix.

Equation 3.4 is applied to each cell in the computational domain and this result in a set of discrete differential equations of the form

$$\frac{\partial \phi}{\partial t} = F(\phi) \quad (3.5a)$$

where the function F incorporates all the spatial discretization of the convective and source terms in equation 3.4.

The cell Reynolds number in the HDD enclosure is defined as

$$\text{Re} = \frac{\rho u d}{\mu}, d = V^{1/3} \tag{3.6}$$

where V is the local cell volume of the enclosure, u is the localized linear velocity, d is defined as cubic root of V , and μ is fluid viscosity.

The steady-state solutions in the results section are achieved by computing the unsteady state flow equations and march in with a local time-step to arrive at a steady-state solution. The contour plot showing the computed Reynolds numbers across plane 1 is illustrated in Figure 3.1. Reynolds numbers in each cell across the selected plane are calculated based on equation 3.6. Plane 1 cut through the HDI above the disk where the largest variation on velocity magnitude occurs and therefore provides a reasonable representation plane. Since the Reynolds numbers across plane 1 is less than 100, the airflow in the HDD enclosure is therefore laminar.

Figure 3.1: Contours of computed cell Reynolds number

The Reynolds number for a HDD enclosure airflow problem can also be calculated by substituting the characteristic length, d of equation 3.6 with the radius of the disk, L . Re-writing Equation 3.6 and replacing the linear velocity with angular velocity, w , gives

$$\text{Re} = \frac{wL^2}{\nu} \tag{3.7}$$

where $\nu = \frac{\mu}{\rho}$, w is the angular velocity of the spinning disk, where $u = wL$, L is an appropriate scaling parameter for length which in this case corresponds to the radius of the disk at 13.5mm, and ν is the kinematic viscosity at $1.4607\text{e-}05 \text{ m}^2/\text{s}$. The calculated Reynolds numbers based on the due to the disk spindle speeds of 3600rpm and 5400rpm are 4704 and 7054 respectively. These low Re numbers imply laminar flows.

The computed Reynolds numbers between laminar & turbulence cases are also verified by comparing both contours plots from the results as illustrated by Figures 3.2(a)-(b). The maximum computed cell Reynolds number for the laminar and turbulence cases are 57.1 and 56.6 respectively. Both contours plots show no significant difference in the computed numbers.

Figure 3.2: Contours of computed cell Reynolds Numbers (a) Laminar flow (b) RSM, turbulent flow

3.1.1. General Scalar Transport Equation

The flow field is solved with a control-volume-based technique to convert a general scalar transport equation to an algebraic equation that are solved numerically. This control volume technique consists of integrating the transport equation about each control volume, yielding a discrete equation that expresses the conservation law on a control-volume basis. Discretization of the governing equations can be illustrated most easily by considering the unsteady conservation equation for transport of a scalar quantity ϕ . The general transport equation as described earlier in Equation 3.4 is written in integral form for an arbitrary volume V as follows;

$$\int_V \frac{\partial \rho \phi}{\partial t} dV + \oint \rho \phi \vec{v} \cdot d\vec{A} - \oint \Gamma_\phi \nabla \phi \cdot d\vec{A} = \int_V S_\phi dV \quad (3.8)$$

where $\nabla \phi$ is gradient of $\phi (= \partial \phi / \partial x) \hat{i} + (\partial \phi / \partial y) \hat{j}$ in 2D), $\vec{A}$ is surface area vector, $\vec{v}$ is velocity vector ($= u \hat{i} + v \hat{j}$ in 2D).

Equation 3.8 is applied to each control volume, or cell, in the computational domain. The two-dimensional, triangular shown in Figure 3.2 is an example of such a control volume. Discretization of equation 3.8 on a given cell yields

$$\frac{\partial \rho \phi}{\partial t} V + \sum_f^{N_{faces}} \rho_f \vec{v}_f \phi_f \cdot \vec{A}_f - \sum_f^{N_{faces}} \Gamma_\phi \nabla \phi_f \cdot \vec{A}_f = S_\phi V \quad (3.9)$$

where N_{faces} is the number of faces enclosing a cell, ϕ_f is the value of ϕ convected through face f , $\rho_f \vec{v}_f \cdot \vec{A}_f$ is the mass flux through the face and A_f is the area of face f $f|A| (= |A_x \hat{i} + A_y \hat{j}|$ in 2D).

Figure 3.3: Control volume used to illustrate discretization of a Scalar Transport Equation

3.1.2 Temporal Discretization

In the transient simulation, the governing equation must be also discretized in both space and time. For the spatial discretization, the discrete values of the scalar ϕ are stored at these cell centers (c_0 and c_1 in Figure 3.3) during the computation. However, face values ϕ_f are required for the convection terms in equation 3.9 and must be interpolated from the cell center values. This is accomplished using an upwind scheme. Upwinding means that the face value ϕ_f is derived from quantities in the cell upstream, or "upwind," relative to the direction of the normal velocity V_n in equation 3.9. In this study, second-order upwind scheme is used to discretize the momentum equation.

The quantities at cell faces are computed using a linear reconstruction approach as described by Barth and Jespersen [56] to achieve second order accuracy. In this approach, a higher-order accuracy is achieved at cell faces through a Taylor series expansion of the cell-centred solution about the cell centroid. Thus, when second-order upwinding is selected, the value ϕ_f is computed using the following expression:

$$\phi_{f,sou} = \phi + \nabla\phi \cdot \vec{r} \quad (3.10)$$

where ϕ and $\nabla\phi$ are the cell-centred value and its gradient in the upstream cell, and $\vec{r}$ is the displacement vector from the upstream cell centroid to the face centroid. This formulation required the determination of the gradient $\nabla\phi$ in each cell. Finally the gradient $\nabla\phi$ is limited so that no new maxima or minima are introduced.

Temporal discretization involves the integration of every term in the differential equations over a time step Δt . This integration of the transient terms is straightforward. The generic expression for the time evolution of a variable ϕ is given by equation 3.5a. The second-order accurate temporal discretization using backward differences is given by

$$\frac{3\phi^{n+1} - 4\phi^n + \phi^{n-1}}{2\Delta t} = F(\phi) \quad (3.11)$$

where ϕ is a scalar quantity, n is the value of the scalar at the current time level, t and Δt is the time step for numerical integration. Once the time derivative has been

discretized, a choice remains for evaluating $F(\phi)$, in particular the time level values of ϕ should be used in evaluating F .

Implicit time integration method is chosen to evaluate $F(\phi)$ at the future time level.

$$\frac{\phi^{n+1} - \phi^n}{\Delta t} = F(\phi^{n+1}) \quad (3.12a)$$

This is referred to as “implicit” integration since ϕ^{n+1} in a given cell is related to ϕ^{n+1} in neighboring cells through $F(\phi^{n+1})$.

$$\phi^{n+1} = \phi^n + \Delta t F(\phi^{n+1}) \quad (3.12b)$$

This implicit equation can be solved iteratively at each time level before moving to the next time step. The implicit scheme is selected due to the unconditionally stable nature with respect to time step size.

3.1.3 Gradients and derivative

Gradients are needed not only for constructing values of a scalar at the cell faces, but also for computing secondary diffusion terms and velocity derivatives. The gradient $\nabla\phi$ of a given variable ϕ is used to discretize the convection and diffusion terms in the flow conservation equations 3.1.

The Green-Gauss theorem is used to compute the gradient of a scalar ϕ at the cell center c_0 and resulting in the following discrete form:

$$(\nabla\phi)_{c_0} = \frac{1}{V} \sum_f \bar{\phi}_f \vec{A}_f \quad (3.13a)$$

where ϕ_f is the value of ϕ at the cell face centroid. The summation is over all the faces enclosing the cell. The face value $\bar{\phi}_f$ is taken from the arithmetic average of the nodal values on the face, using the Green-Gauss Node-based gradient evaluation, i.e.,

$$\bar{\phi}_f = \frac{1}{N_f} \sum_n^{N_f} \bar{\phi}_n \quad (3.13b)$$

where N_f is the number of nodes on the face.

The nodal values, $\bar{\phi}_n$ in equation 3.13b, are constructed from the weighted average of the cell values surrounding the nodes, following the approach originally proposed by Holmes and Connel [57] and Rauch *et. al.* [58]. This scheme reconstructs exact values of a linear function at a node from surrounding cell-centred values on arbitrary unstructured meshes by solving a constrained minimization problem, preserving a second-order spatial accuracy. The node-based averaging scheme is known to be more accurate than the default cell-based scheme for unstructured meshes, most notably for triangular and tetrahedral meshes.

3.1.4 Pressure-Based Solver

The discretization scheme described in Section 3.1.2 for a scalar transport equation is also used to discretized the momentum equations. In this section, the discretization of the momentum and continuity equation by means of a pressure-based solver, are addressed. The continuity and momentum equations in integral form are given by:

$$\frac{\partial}{\partial t} \int_V \rho dV + \oint \rho \vec{v} \cdot d\vec{A} = 0 \quad (3.14a)$$

$$\frac{\partial}{\partial t} \int_V \rho \vec{v} dV + \oint \rho \vec{v} \vec{v} \cdot d\vec{A} = - \oint p I \cdot d\vec{A} + \oint \vec{\tau} \cdot d\vec{A} + \int_V \vec{F} dV \quad (3.14b)$$

where I is the identity matrix, $\vec{\tau}$ is the stress tensor, and $\vec{F}$ is the force vector.

The x-momentum equation can be obtained by setting $\phi = u$.

$$\rho p u = \sum_{nb} a_{nb} u_{nb} + \sum p_f A \cdot \hat{i} + S \quad (3.14c)$$

However, the velocity field cannot be obtained, as the pressure field and face mass flux are not known a priori. A co-located scheme is adopted whereby pressure and velocity are both stored at cell centers. However, Equation 3.14c requires the value of the pressure at the face between cell c0 and c1, shown in Figure 2.27. Therefore, an interpolation scheme is required to compute the face values of pressure from the cell

values. The pressure values at the faces use momentum equation coefficients as outlined by Rhie and Chow [59].

$$P_f = \frac{\frac{P_{c0}}{a_{p,c0}} + \frac{P_{c1}}{a_{p,c1}}}{\frac{1}{a_{p,c0}} + \frac{1}{a_{p,c1}}} \quad (3.14d)$$

Equation 3.14d works well when the pressure variation between cell centers is smooth. When there are jumps or large gradients in the momentum source terms between control volumes, the pressure profile has a high gradient at the cell face, and cannot be interpolated using this scheme. The resultant of large pressure jump profile is discrepancy shows up in overshoots/undershoots of cell velocity.

The standard scheme is used to interpolate the pressure values at the faces using momentum equation coefficients as given by Rhie and Chow [59]. The standard scheme is selected because the computational domain has relatively low body-forces due to a low speed rotating flow.

The continuity Equation 3.14a in integral form may be integrated over the control volume in Figure 2.27 to yield the following discrete equation.

$$\sum_f^{N_{faces}} J_f A_f = 0 \quad (3.14e)$$

where J_f is the mass flux through face f , ρv_n .

In order to proceed further, it is necessary to relate the face values of velocity, $\vec{v}_n$ to the stored values of velocity at the cell centers. The face value of velocity is not averaged linearly, instead momentum-weighted averaging, using weighting factors based on the a_p coefficient from equation 3.14c is performed. Using this procedure, the face flux, J_f , may be written as

$$\begin{aligned}
J_f &= \rho_f \frac{a_{p,c0} v_{n,c0} + a_{p,c1} v_{n,c1}}{a_{p,c0} + a_{p,c1}} + d_f ((p_{c0} + (\nabla p)_{c0} \cdot \vec{r}_0) - (p_{c1} + (\nabla p)_{c1} \cdot \vec{r}_1)) \\
&= \hat{J}_f + d_{f(p_{c0}-p_{c1})}
\end{aligned} \tag{3.14f}$$

where $p_{c,0}$, $p_{c,1}$, and $v_{n,c0}$, $v_{n,c1}$ are the pressures and normal velocities, respectively, within the two cells on either side of the face, and $\hat{J}_f$ contains the influence of velocities in these cells. The term d_f is a function of $\bar{a}_p$, the average of the momentum equation a_p coefficients for the cells on either side of face f .

After acquiring of pressure and velocity from the equations, the use of a suitable scheme for pressure-velocity coupling is important with the segregated solver where the momentum and continuity equations are solved at each step discretely. The Pressure-Implicit with Splitting of Operator (PISO) pressure-velocity coupling scheme, with neighbor correction is highly recommended for transient flow calculations and was chosen for this work. PISO can maintain a stable calculation with a large time step and an under-relaxation factor of 1.0 for both momentum and pressure. PISO is also tested with highly skewed mesh and prove to handle convergence easier as compared to other scheme (SIMPLE, SIMPLEC, etc.)

The PISO algorithm however takes a little more CPU time per solver iteration, but it can dramatically decrease the number of iterations required for convergence, especially for transient problems. For meshes with a high degree of skewness, the simultaneous coupling of the neighbor and skewness corrections at the same pressure correction equation source may cause divergence or a lack of robustness. An alternate, although more expensive, method for handling the neighbor and skewness corrections inside the PISO algorithm is to apply one or more iterations of skewness correction for each separate iteration of neighbor correction. For each individual iteration of the classical PISO algorithm, this technique allows a more accurate adjustment of the face mass flux correction according to the normal pressure correction gradient.

3.1.5 Boundary conditions on global flow field

Wall boundary conditions are used to bound fluid region. The non-slip boundary condition is enforced for every wall in the HDD domain. The induced momentum to the rotating flow in the HDD enclosure is created by setting a tangential velocity component in terms of a rotational motion of the wall boundary defined by the disk as shown in Figure 1.2. A "non-slip" model wall with zero shear is also generalized for all the rigid wall boundaries.

The energy equation as described by Equation 3.3 is solved by defining thermal boundary conditions at the rotating motor-hub wall as shown by Figure 1.2. The motor-hub wall is set as a zero-thickness wall and the wall thermal resistance and heat generation are assumed insignificant.

The thermal condition for the motor-hub wall is set as a fixed temperature wall. The specified temperature at the wall surface is 333k to simulate motor-coil heating at steady state condition after long operation hours. The heat transfer to the wall is computed using the following equation;

$$q = h_f (T_w - T_f) + q_{rad} \quad (3.15)$$

where h_f = fluid-side local heat transfer coefficient, T_w = wall surface temperature, T_f = local fluid temperature, and q_{rad} = radiative heat flux.

The re-circulation filter in the HDD is modeled as a porous media. The porous media model in *Fluent* can be used for a wide variety of problems, including flows through packed beds, filter papers, perforated plates, flow distributors, and tube banks. Heat transfer through the medium can also be represented, subject to the assumption of thermal equilibrium between the medium and the fluid flow. However, heat through the re-circulation filter is not significant and the porous media model can therefore be further simplified to a one-dimensional problem, termed as the "porous jump".

The filter is defined as a thin membrane with known velocity/pressure-drop. The porous jump model is applied to a face zone of the filter, and the pressure loss in the flow is determined via inputs. This one-dimensional model is used (instead of the full porous media model) because of the higher robustness and ease of convergence.

The thin porous medium has a finite thickness over which the pressure change is defined as a combination of Darcy's Law and an additional inertial loss term:

$$\Delta p = -\left(\frac{\mu}{\alpha} v + C_2 \frac{1}{2} \rho v^2\right) \Delta m \quad (3.16)$$

where μ is the laminar fluid viscosity, α is the permeability of the medium, C_2 is the pressure-jump coefficient, v is the velocity normal to the filter face, and Δm is the thickness of the medium. α is defined as $7\text{e-}10\text{m}^2$ and C_2 is set as 0.2. The porous media thickness is defined as 0.001m.

3.2 Discrete Phase Model (DPM) for computing trajectories

In this section, the particle transport model is introduced. The objective for tracking the trajectories of particles in the HDD enclosure provides a mean to understand the air induced failure mechanism caused to the slider in the event of a head crash. The pre-occurrence event in a head crash or disk scratch has significant connection with the particles trajectories which is influenced greatly by the surrounding wall shapes interaction in a small volume. By coupling the CFD techniques discussed earlier with a suitable particle transport model, the tracking of sub-micron sized particle can be achieved. The discrete phase model (DPM) is written in a Lagrangian reference frame and predicts the trajectory of discrete phase particles by means of integrating equations of motion obtained with the balance of the inertial forces acting on the particle and resisted by the forces due to the various effects of drag, gravity, heat and lift, etc. The governing equation can be simply written (for the x direction in Cartesian coordinates) as

$$\frac{du_p}{dt} = F_D(u - u_p) + \frac{g_x(\rho_p - \rho)}{\rho_p} + F_x \quad (3.16)$$

where F_D is defined the drag force per unit particle mass and F_x is an additional acceleration term. Here u is the fluid phase velocity, u_p is the particle velocity, ρ is the fluid density, ρ_p is the density of the particle, and μ is the molecular viscosity of the fluid. For a rigid sphere moving in an air-flow, the drag force can be expressed as

$$F_D = \frac{18\mu}{\rho_p d_p^2} \frac{C_d \text{Re}}{24} \quad (3.17)$$

where C_d is the drag coefficient, d_p is the diameter of the sphere. Re is the relative Reynolds number, which is defined as

$$\text{Re} \equiv \frac{\rho d_p |u_p - u|}{\mu} \quad (3.17a)$$

The drag coefficient C_d , can be taken from either

$$C_d = a_1 + \frac{a_2}{\text{Re}} + \frac{a_3}{\text{Re}^2} \quad (3.17b)$$

where a_1, a_2, a_3 are constants that apply to smooth spherical particles over several ranges of Re given by Morsi and Alexander [60] or

$$C_d = \frac{24}{\text{Re}_{sph}} (1 + b_1 \text{Re}_{sph}^{b_2}) + \frac{b_3 \text{Re}_{sph}}{b_4 + \text{Re}_{sph}} \quad (3.17c)$$

where $b_1 = \exp(2.3288 - 6.4581\phi + 2.4486\phi^2)$, $b_2 = 0.0964 + 0.5565\phi$,
 $b_3 = \exp(4.905 - 13.8944\phi + 18.4222\phi^2 - 10.2599\phi^3)$,
 $b_4 = \exp(1.4681 - 12.2584\phi + 20.7322\phi^2 + 15.8855\phi^3)$, which is taken from Haider and Levenspiel [61].

The shape factor, ϕ , is defined as

$$\phi = \frac{s}{S} \quad (3.17d)$$

where s is the surface area of a sphere having the same volume as the particle, and S is the actual surface area of the particle. The Reynolds number Re_{sph} is computed with the diameter of a sphere having the same volume.

For very small particles, some other researchers use the theory of gas dynamics approach for calculating C_d . Liu *et al.* [62] studied the drag of a sphere in a flow of a rarefield gas by using the Boltzmann equation for Maxwellian molecules. They obtained the result that is a modification of the result of a free molecule flow. However, in this model, a

form of Stoke's drag law as stated by Ounis *et al.* [63] is used instead in correlation with sub-micron size particles as defined by

$$F_D = \frac{18\mu}{d_p^2 \rho_p C_c} \quad (3.18)$$

The C_c is the Cunningham correction factor can also be computed as

$$C_c = 1 + \frac{2\lambda}{d_p} \left(1.257 + 0.4e^{-(1.1d_p/2\lambda)} \right) \quad (3.18a)$$

where λ is the molecular mean free path.

The second term in the right hand side of equation (3.2a), $\frac{g_x(\rho_p - \rho)}{\rho_p}$, etc. is the net force due to gravity and up thrust per unit mass on the particle. Gravity force is related to the occupied space by the particle itself (In this case, sphere is assumed). The density, ρ , of air & particle ρ_p defined by Alumina (Al_2O_3) is 1.225kg/m^3 and 3980 kg/m^3 at ambient condition respectively. Since $\rho \ll \rho_p$, the gravity term becomes $m_p \times g_x$, where g_x is in order of 1 and the entire term is governed by the mass which is dependent on the volume. For d_p in the sub-micron range, gravity force is insignificant.

Equation 3.15 also incorporates additional forces F , which may act on the particle in special circumstances. Typical forces that are considered in this investigation include forces due to pressure gradient, thermophoretic force, Brownian force and Saffman Lift force. In this study, Magnus lift force, which is the effect of the spinning of the sphere in a fluid, is omitted in the calculation. Magnus lift force is less by an order of magnitude than Saffman lift forces unless the rotating speed is very much larger than the rate of shear. Since the rotation of a sphere is determined by the torque acting on it by the viscous flow and is unknown before calculation, it is necessary to solve to acquire the knowledge of the intensity therefore significant in the governing equation.

3.2.1 Saffman Lift Force

When a rigid sphere moves in a shear flow, it is acted upon by a lift force, which is perpendicular to the flow direction of the fluid at which the sphere is located. A generalization of the equation has also been outlined by Saffman [64] and it is given by

$$\vec{F} = \frac{2K\nu^{1/2}\rho d_{ij}}{\rho_p d_p (d_{ik}d_{kl})^{1/4}} (\vec{u} - \vec{u}_p) \quad (3.19)$$

where $K = 2.594$ and d_{ij} is the deformation tensor.

This form of the lift force is intended for small particle Reynolds numbers. Also, the particle Reynolds number based on the particle-fluid velocity difference must be smaller than the square root of the particle Reynolds number based on the shear field. The velocity in the HDD enclosure decrease in the Z-direction, thus the Saffman force will point to the HDD top cover when the assumed spherical particle moves faster than the airflow at the position where it is located. As the Saffman lift force is only valid for submicron particles, it is therefore only applied to specify cases when diameter is less than a micron.

3.2.2 Pressure Gradient Force

One additional force F is the “virtual mass” force, i.e. the force required to accelerate the fluid surrounding the particle. This force can be written as

$$\vec{F} = \frac{1}{2} \frac{\rho}{\rho_p} \frac{d}{dt} (\vec{u} - \vec{u}_p) \quad (3.20)$$

and it is not significant when alumina (Al_2O_3) particles is considered as $\rho > \rho_p$. The additional force, which arises due to the pressure gradient in the fluid, is defined as:

$$\vec{F} = \left(\frac{\rho}{\rho_p} \right) \vec{u}_p \frac{\partial \vec{u}}{\partial \vec{r}} \quad (3.21)$$

where $\vec{r} = (x, y, z)$. This pressure gradient force is also not significant in the assessment of alumina (Al_2O_3) particles, which has a density much higher than air.

3.2.3 Thermophoretic Force

Small particles suspended in a gas that has a temperature gradient experience a force in the direction opposite to that of the temperature gradient. This phenomenon is known as thermophoresis. The thermophoretic force acting on the particles can be deduced from the equation as suggested by Talbot [65]:

$$\vec{F} = -\frac{6\pi d_p \mu^2 C_s (K + C_t Kn)}{\rho(1 + 3C_m Kn)(1 + 2K + 2C_t Kn)} \frac{1}{m_p T} (\nabla \cdot T) \quad (3.22a)$$

where $D_{T,p}$ is the thermophoretic coefficient given by $\frac{6\pi d_p \mu^2 C_s (K + C_t Kn)}{\rho(1 + 3C_m Kn)(1 + 2K + 2C_t Kn)}$ and equation 3.21a can be simplified to;

$$\vec{F} = -D_{T,p} \frac{1}{m_p T} (\nabla \cdot T) \quad (3.22b)$$

where: Kn = Knudsen number = $2\lambda/d_p$, $K = k/k_p$, k = fluid thermal conductivity based on translational energy only = $(15/4) \mu R$, k_p = particle thermal conductivity, $C_s = 1.17$, $C_t = 2.18$, $C_m = 1.14$, m_p = particle mass, T = local fluid temperature, and μ = fluid viscosity.

The effect of thermophoretic force is significant in the heat dissipation zone near the rotorhub wall, which is given by the spindle motor coil. A more extensive discussion would be made in the result section due to thermophoretic force effect in the HDD enclosure.

3.2.4 Brownian Force

For sub-micron particles, the effects of Brownian motion can be optionally included in the additional force term. When a small particle is suspended in a fluid, it is subjected to the impact of gas or liquid molecules. For ultra fine particles (colloids), the instantaneous momentum imparted to the particle varies randomly which causes the particle to move on an erratic path resulting in what is known as Brownian motion. Figure 3.4 illustrates the Brownian motion process

Figure 3.4: Brownian motion

In the present work, the components of the Brownian force are modeled as a Gaussian white noise process with spectral intensity $S_{n,ij}$ given by Li and Ahmadi [66] as

$$S_{n,ij} = \frac{216\nu\sigma T}{\pi^2 \rho d_p^5 \left(\rho_p/\rho\right)^2 C_C} \delta_{ij} \quad (3.23)$$

where T is the absolute temperature of the fluid, ν is the kinematic viscosity, and K_B is the Boltzmann constant. Amplitudes of the Brownian force components are of the form

$$F_{bi} = -\zeta i \sqrt{\frac{\pi S_{n,ij}}{\Delta T}} \quad (3.24)$$

3.2.5 Forces in Rotating Reference Frames

One important force term, F , in equation 3.15 is the forces on particles that arise due to rotation of the disk in the HDD enclosure. These forces arise when modeling flows in rotating frames of reference is an essential component. For this problem, the rotation is defined about the Z axis, for example, the forces on the particles in the Cartesian X and Y directions can be written as

$$\left(1 - \frac{\rho}{\rho_p}\right) \Omega^2 x + 2\Omega \left(u_{y,p} - \frac{\rho}{\rho_p} u_y\right) \quad (3.25)$$

where $u_{y,p}$ and u_y are the particle and fluid velocities in the Cartesian y direction, and

$$\left(1 - \frac{\rho}{\rho_p}\right) \Omega^2 y + 2\Omega \left(u_{x,p} - \frac{\rho}{\rho_p} u_x\right) \quad (3.26)$$

where $u_{x,p}$ and u_x are the particle and fluid velocities in the Cartesian x direction.

3.2.6 Boundary Conditions

When a particle impacts onto a boundary surface in the enclosure, the outcome of the impact on the particle can have several possibilities. Four possibilities are catered for this simulation as follows: a. particle may reflect via an elastic or inelastic collision, b. particle may escape through the boundary. (The particle is then lost from the calculation at the point where it impacts the boundary.), c. particle may get trapped at the wall. Non-volatile material is lost from the calculation at the point of impact with the boundary; volatile material present in the particle or droplet is released to the vapor phase at this point, and lastly, d. particle may pass through an internal boundary zone, such as radiator or porous jump.

These possibilities are sketched schematically in Figure 3.4. In the HDD model case, a trapped wall is assigned to the spinning disk and anywhere else is set as a reflective boundary.

Figure 3.5: Possible boundary condition upon particles impact

3.3 Fine Particle Model (FPM) for air-field representation

The other transport model uses the classical Eulerian reference frame of tracking and treat the particles in terms of their size distribution, $n(m_p, \vec{x}, t)$ where m_p represent particle mass. The code is named the Fine Particle Model (FPM) in *Fluent* and looks into the effect of particle-to-particle interaction coupling with the molecular forces. The FPM is used to characterize particle contamination in the HDD enclosure using the aerosol dynamics.

Lagrangian methods are often the most efficient used to sample a fluid domain and the physical conservation laws begin with a Lagrangian perspective. Nevertheless, almost all of the theory in fluid dynamics is developed in the Eulerian or field form. The transformation of the conservation laws from a Lagrangian to an Eulerian system requires three key results. 1) The first is dubbed the Fundamental Principle of Kinematics; the velocity at a given position and time (the Eulerian velocity) is equal to the velocity of the particle that occupies that position at that time (the Lagrangian velocity). 2) The material time derivative relates the time rate of change observed following a moving parcel to the time rate of change and advective rate of change observed at a fixed position; $\frac{D(\cdot)}{Dt} = \frac{\partial(\cdot)}{\partial t} + \vec{v} \cdot \nabla(\cdot)$. 3) The time derivative of an integral over a moving fluid volume can be transformed into field coordinates by means of the Reynold Transport Theorem.

In the FPM, the flow and interactions of particles are describe by the general dynamic equation as in Friedlander [67] and which is expressed as

$$\frac{\partial n}{\partial t} + \nabla \cdot n \vec{U} = \left[\frac{\partial n}{\partial t} \right]_{diff} + \left[\frac{\partial n}{\partial t} \right]_{coag} + \left[\frac{\partial n}{\partial t} \right]_{ext} \quad (3.27)$$

Unlike the Lagrangian technique which treats Brownian motion as a force as describe in Section 3.2, the Brownian motion here is treated here as a part of diffusion term, $\left[\frac{\partial n}{\partial t} \right]_{diff}$, which can be written as $\nabla \cdot (\rho D_{PM} \nabla \frac{n}{p})$ with D_{PM} as the Brownian diffusivity. Hesketh [68] defined the Brownian diffusivity within the transition regime as

$$D_{PM} = \frac{kT}{3\pi\eta d_p} C_c \quad (3.27a)$$

where k is the Boltzmann constant. The coagulation term, $\left[\frac{\partial n}{\partial t}\right]_{coag}$, takes into account the particle-particle collisions and interactions and can be written as

$$\left[\frac{\partial n}{\partial t}\right]_{diff} = \frac{1}{2} \int_0^{m_p+m_p'} K_c \cdot n(m)n(m_p')dm_p - n(m_p+m_p') \int_0^\infty K_c \cdot n(m_p)dm_p \quad (3.27b)$$

where the coagulation kernel, K_c can be expressed as

$$K_c = f_{trans} \cdot 2\pi(d_p + d_p')(D_{PM} + D_{PM}') \quad (3.27c)$$

with $f_{trans} = \frac{1 + Kn_c}{1 + 2Kn_c(1 + Kn_c)}$ as defined by Dahneke¹⁸ and

$$Kn_c = \frac{4(D_{PM} + D_{PM}')}{(d_p + d_p')\sqrt{u(m_p)^2 + u(m_p')^2}}; \quad u(m_p) = \sqrt{\frac{8kT}{\pi m_p}} \quad (3.27d)$$

The term due to the external forces, $\left[\frac{\partial n}{\partial t}\right]_{ext}$ considers other types of sedimentation, thermophoresis, and electro-magnetism.

The transport equation for the aerosol dynamics i.e. equation (3.3f) is solved using Fine Particle Model (FPM) developed by Whitby *et.al* [69] which uses model aerosol dynamics technique described in Whitby [70]. In this technique, the size distribution function $n_j(m_p)$, used in the FPM is the lognormal distribution. Equation (3.3f) is then solved using the “method of moments” in Whitby [70] where the moment of each mode can be written as

$$M_{j,k} = \int_0^\infty m_p^k n_j(m_p)dm_p \quad (3.27e)$$

In order to establish an equivalent comparison with the discrete phase model (DPM), the coagulation term, which enables the formation of larger particles or the breakdown into smaller particles due to collision, is neglected to ensure that the particle-particle interaction in the HDD enclosure is ignored. Identical boundary conditions are used for the disk & casing surfaces, implying that the particles would be “trapped” upon contact with any of these surfaces. The external force considered in this investigation primarily comes from sedimentation but thermophoresis effects would also be evaluated as well subsequently.

CHAPTER 4

RESULTS AND DISCUSSIONS

The flow generated by the spinning disk in a typical one-inch HDD enclosure is computationally simulated. The computational results are attained using a simple model and a geometrically true model. The superior results, in term of resolution and accuracy extracted from the detailed model are compared to a simple model. The flow fields are computed by solving the unsteady-state NS equation until the residual converged to attain the steady-state results. Grid convergence test is performed on the computational mesh to eliminate errors due to mesh quality variation. The model convergence is dependent on the Grid Convergence Index that is based upon a grid refinement error estimator derived from the theory of generalized Richardson Extrapolation. Arm positional effects on flow distribution are also analyzed by rotating the arm about the pivot to three different angles reference to the ID, MD, and OD positions on the disk. The generated grids with the various arm positions are solved to study the flow responses to each arm position changes. The grids from selected regions of interest are further enhanced using solution-adaptive refinement. This method enable cells to be added where needed, thus enabling the features of the flow field to be better resolved.

Grid convergence test is performed on the computational mesh to eliminate errors due to mesh quality variation. The model convergence is dependent on the Grid Convergence Index that is based upon a grid refinement error estimator derived from the theory of generalized Richardson Extrapolation. Arm positional effects on flow distribution are also analyzed by rotating the arm about the pivot to three different angles reference to the ID, MD, and OD positions on the disk. The generated grids with the various arm positions are solved to study the flow responses to each arm position changes. The grids from selected regions of interest are further enhanced using solution-adaptive refinement. This method enable cells to be added where needed, thus enabling the features of the flow field to be better resolved.

Thermal effects are crucial in the analysis of HDD airflow since the variation of temperature can be significant. Heat generated by components inside the HDD enclosure produces a thermophoretic force, which influences thermal conductive particle trajectories and affects flow distribution. This is modeled by a constant heating motor-hub wall that defines the heat input into the enclosure. The momentum and energy equation are solved respectively to acquire the computed velocity and temperature field. Based on the acquired steady airflow patterns, particle trajectories are then tracked using the Discrete Phase Model (DPM) available in *Fluent* that solves the particle inertia force balance equation with the NS transport equation. Particles of different sizes and materials are injected at various locations in the computed flow field and the corresponding particle trajectories are analyzed. The effect of particle-to-particle interaction is also studied by using an alternative model, Fine Particle Model (FPM) for predicting particles diffusion as well as interactions through collision and coagulation based on an Eulerian frame of reference. The particle concentration and size distribution provide good indication of aerosol behavior within the HDD enclosure.

4.1 COMPUTED AIRFLOW CHARACTERISTICS IN A BASIC HDD ENCLOSURE

The basic computational model of an one-inch small form factor HDD as illustrated by Figures 2.11-2.13 is used for this computation. The simplicity of this basic model as described in Chapter 2, offers the benefits of lower computational effort, scalability for unstructured meshes and robustness of solution convergence. The computed result provides a good comparative platform for the complex geometrical model and is a useful tool for the preliminary analysis of the HDD airflow problem, as well as a medium for acquiring a quicker result.

The basic HDD enclosure uses a generalized mesh model that includes only essential flow influenced components (disk, etc) that drives the airflow and a selective of elements with highly simplified geometry. The details on the complex suspension is omitted and substituted by a general beam structure that has a constant thickness and a tapered converging extension.

The meshing of the HDD enclosure space is relatively simple. The main step involves with the generation of a fitted three-dimensional grid of tetrahedral elements on the interior of the HDD enclosure. The CAD model is treated as a single entity without decomposing into smaller volumes while still retaining good qualities of the volume meshes. This is achievable with the strategy of developing generally good quality surface meshes and mesh refinement at close gap region through vertex smoothing. The final step is completed with volume mesh function by selecting an appropriate size distribution. Area where smaller gap exists would be automatically taken care by the preprocessed surface elements. The control of final mesh qualities is therefore highly dependent on the surrounding surface mesh distribution where they are referenced for growth.

The entire volume is meshed using the Tet/Hybrid scheme with *Gambit*. This scheme composes primarily of hexahedral elements but also includes wedge elements where appropriately assessed by the meshing software. Wedge elements are usually added at the boundaries to fill gaps where the regular grid of hexahedral does not flushes with the surface. As the geometry is highly asymmetrical along each z-plane, it would be very time-consuming and unfeasible to mesh structurally using structured mesh without further decomposing the volume. Unstructured meshing is also preferred as the flow is not unidirectional in all parts of the HDD enclosure.

To validate the mesh qualities for this computation, the examination of the spatial convergence is necessary. This examination is a straightforward method for determining the ordered discretization error in a typical CFD simulation. It involves performing the simulation on two or more successively finer grids.

Roache *et al.* [71,72] proposes a grid convergence index (GCI) to provide a consistent manner for reporting the results of grid convergence studies and perhaps provide an error band on the grid convergence of the solution. The GCI can be computed using two levels of grid; however, three levels are recommended in order to accurately estimate the order of convergence and to check that the solutions are within the asymptotic range of convergence. The GCI is a measure of the percentage the computed

value is away from the value of the asymptotic numerical value. It indicates an error band on how far the solution is from the asymptotic value. It also indicates how much the solution would change with a further refinement of the grid. The GCI is based upon a grid refinement error estimator derived from the theory of generalized Richardson Extrapolation. It is recommended for use whether or not Richardson Extrapolation is actually used to improve the accuracy, and in some cases even if the conditions for the theory do not strictly hold.

The easiest approach for generating the series of grids is to generate a grid with what one would consider fine grid spacing. Then coarser grids are obtained by removing every other vertex node on grid lines in each coordinate direction across the entire model. This approach is continued to create additional levels of coarser grids.

Non-integer grid coarsening is used in this grid coarsening. This is desired as halving a grid may put the solution out of the asymptotic range. Non-integer grid refinement or coarsening will require the generation of a new grid. It is important to maintain the same grid generation parameters as the original grid. The approach is to select several grid spacing as reference grid spacing. The grid spacing in *Gambit* can be defined by two ways, either by interval count or interval size. Interval count puts the actual number of mesh intervals to be placed on the edge, which is difficult to maintain equal nodes intervals along varying edge lengths. On the contrary, interval size defines an equal distribution of nodes on the edge according to the following equation: $n = L / d$, where n is the number of intervals on the edge, L is the edge length, and d is the interval size (user input).

Interval size of 0.3, 0.5 and 1.0 are selected as reference grid spacing. For the three varying grid models, the arm component and the parallel surface above and under the HDI are pre-meshed and perform refinement whenever necessary. The control for these surface meshes make use of controlled vertex spacing assigned to edges. These surfaces are essential to be pre-meshed due to the existing small gap spacing attributed by geometrical shapes and placement. By applying a general mesh interval size on the entire enclosure would result in high skewed mesh volumes and large gradient size jump. Upon

the meshing of the selected entities, volume mesh function can be activated for all other surfaces without overwriting the customized surfaces. Interval size of 0.3 is the smallest grid spacing that the software can handle the volume to produce 2.3 millions elements. Increasing the grid size spacing performs subsequent coarsening to the mesh. Physical inspection on the final model is necessary to ensue homogenous refinement or coarsening in the spatial distribution.

When the grid is refined (grid cells become smaller and the number of cells in the flow domain increase) and the time step is refined (reduced), the spatial and temporal discretization errors, respectively, should asymptotically approaches zero, excluding computer round-off errors. Grid convergence study is then performed with the computed results from these mesh domains.

Table 1 shows the grid convergence table, illustrating the maximum magnitudes of two selected computed flow parameters. The arm is also positioned at the OD for all the different size grids to retain consistency. The maximum velocities and pressures listed are computed from three successive grid sizes meshes, given by G1, G2 and G3.

G1 corresponds to the finest size grid that contains 2.3 millions tetrahedral elements. G2 and G3 have 1.4 and 0.8 millions elements respectively. The grid refinement ratio, R_m , defined by ratio of number of elements for any grid, divided over by the number of elements of the coarsest grid, G3 is presented in Table 1. R_m is used to determine if discretization error can be differentiated from other error sources like iterative convergence errors or computer round off errors, etc. R_m must be above 1.5 to enable the identification of discretization errors and therefore G1 and G2 have met the requirement.

	Grids (Millions)	Grid Refinement Ratio, R_m	Max Velocity (m/s)	Max Pressure (Pa)
G1	2.3	1.6	4.1322	33.557
G2	1.4	1.75	4.405	33.33
G3	0.8	1	4.5045	35.461

Table 1: Grid convergence table

An approach to evaluate the order of convergence using a constant grid refinement ratio r ,

$$P = \frac{1}{\ln(r_m)} \ln \left[\frac{f_3 - f_2}{f_2 - f_1} \right] \quad (5.1)$$

where r_m = grid refinement ratio, f = measured flow parameters.

The lower of the two acceptable r_m is used and defined value here is 1.75, while maximum velocity $V_{\max}$ is selected here to represent f . The computed P value with the inputs is 1.9, which defines the observed order of convergence to approximately 2. The computed order of convergence implies that the solution agreed well with the 2nd order discretization scheme to be used for all the computations.

The GCI on a fine grid is defined as

$$GCI_{fine} = \frac{F_s |\varepsilon|}{r_m^p - 1} \quad (5.2)$$

where F_s is a factor of safety. The refinement may be spatial or in time. The factor of safety is recommended to be $F_s = 1.25$ for comparisons over three or more grids. The higher factor of safety is recommended for reporting purposes and is quite conservative of the actual errors. The relative error ε is defined as,

$$\varepsilon = \frac{f_2 - f_1}{f_1} \quad (5.3)$$

where $f_1 = V_{\max 1} = 4.1322 \text{ m/s}$ and $f_2 = V_{\max 1} = 4.405 \text{ m/s}$, therefore $\varepsilon = 0.0619\%$. Therefore by substituting the calculated ε value into equation 5.2, the computed GCI for G2 gives 0.0445. A small value of GCI indicates that the computation is within the asymptotic range and the grids density is sufficiently fine to results in computational errors due to grid refinement.

The values of the computed flow parameters on Table 1 are also plotted into graphs for better illustration purpose. Figures 4.1(a)-(b) shows the convergence plots with the maximum velocity and pressure (gauge) computed from the three different grids. The horizontal axis plots the reciprocal of the number of grid elements. The vertical axis of Figure 4.2(a) plots the maximum computed velocity against each grid number while Figure 4.2(b) plots the maximum (gauge) pressure. Both plots show significant variation in gradient change between grids G1 and G2 correspond to 0.8 and 1.3 on the horizontal axis (1.4 and 0.8 million elements respectively). From this two plots, it can be observed that a minimum grid number of 1.4 millions should used in the computational domain so that solutions variation between cases do not contribute due to grids variation. Based on an arbitrary safety factor of 1.4, the minimum number of grids should be maintained above 2 millions for any case.

The setup of the following simulations is illustrated with Figure 4.2. The arm is stationary while the disk is set to spin at 3600rpm in a counter-clockwise (CCK) direction. The air is set to rotate by the inertia force on the fluid parcels induced by the spinning disk that is rigidly secured to the rotor-hub. The disk measures $\phi 25.4\text{mm}$ (1.0in) in diameter and has a thickness of 0.381mm (15 μin). The rotor-hub wall is set constant temperature wall of 333K (60°C). This is a time-averaged value obtained experimentally by the design center using a Digital Infrared Thermal Imaging scanner across the HDD for a drive that operates for up to an hour of operation. This value quantify the skin surface temperature at the motor hub and responsible for the advection part in the enclosure. The wall thickness of the motor hub is in the sub-millimeter range and effect of heat conduction through wall is not accounted into the calculation. Figure 4.3 shows the motor hub and the direction of heat diffusion releasing into the enclosure.

All the surfaces are set as rigid wall, with “non-slip” condition. The effect of gravity is included into the governing equation along the negative Z direction, with an acceleration of 9.81m/s^2 . Air density is set as a constant value of 1.225 kg/m^3 at ambient condition. The specific heat of air at constant pressure, C_p is 1006.43 J/kg-k , and the thermal conductivity is defined by 0.0242w/m-k . Viscosity of air at room temperature is $1.7894\text{e-}05\text{kg/m-s}$.

The method of choosing an appropriate time-step size Δt for this problem is by observing the number of iterations required to converge at each time step. The ideal number of iterations per time step is between 5 to 10. If the number of iterations needs substantially more than 10 steps, the time step assigned will be too large. A small time-step Δt of $10e^{-07}$ sec is used for the first 20 time steps and changes to a bigger Δt of $10e^{-06}$ sec to complete the calculation. It is observed that the time-dependent calculation has a very fast "startup" transient that decays rapidly. Therefore, a conservatively small Δt is chosen for the first 5-10 time steps and then gradually increased as the calculation proceeds. To model the transient phenomena properly, the time step Δt is set at least one order of magnitude smaller than the smallest time constant in the HDD enclosure being modeled.

A plot of the Courant number in the enclosure is presented in Figure 4.4 across various planes. The Courant number is given by:

$$CFL = \frac{\overline{v} \Delta t}{\Delta x_{cell}} \quad (5.4)$$

where CFL = Courant number, Δx_{cell} = dimension of the grid cell at each location, $\overline{v}$ = average linear velocity at that location, and Δt is the maximum time step size.

The Courant number reflects the portion of a cell that an air parcel will traverse by advection in one time step and should not exceed a value of 20-40 in most sensitive transient regions of the domain.

The various visualization planes that are extensively used in this chapter are illustrated with Figures 4.4-4.6. Figure 4.4(a) shows *plane 1*, the mid-height X-Y plane between the top disk surface and the top slider. Figure 4.4(b) shows a similar plane 2 with a different z-height elevation. Plane 2 cuts between the bottom slider and bottom disk surface. Figure 4.5(a) shows plane 3, which cuts through the rotating axis of the motor-hub, parallel along Y-Z plane. Figure 4.5(b) shows a similar plane 4, normal to plane 3, cutting the rotating axis of the motor-hub. Plane 5 is illustrated in Figure 4.6(a)

that cuts along the arm axis at the midsection of the structure. Similarly, plane 4.6(b) shows the mid-section of the ramp along the axis of the slope. Figure 4.7(a)-(c) shows the top view of the basic HDD enclosure model used in the study with the different arm positions (ID, MD and OD).

(a)

(b)

Figure 4.1: Convergence plots based on (a) maximum velocity (b) maximum total pressure

Figure 4.2: Arm and Disk assembly showing disk rotation direction and the position of the arm at Inner Diameter.

Figure 4.3: Z-X view on arm assembly, showing heat dissipation from the rotor-hub wall

Figure 4.4 – Visualization (a) Plane 1: Interval between top slider and disk (b) Plane 2: Interval between bottom slider and disk.

Figure 4.5 – Visualization (a) Plane 3: Y-Z plane of hub midsection (b) Plane 4 : X-Z plane of hub midsection.

Figure 4.6 – Visualization (a) Plane 5: Midsection along the arm axis (b) Plane 6: Midsection along the ramp axis

(a)

(b)

(c)

Figure 4.7: Top view of Simple model and arm at (a) ID (b) MD (c) OD

Flows in a real disk drive enclosure are highly unsteady with largely laminar regions in the 1-inch small form factor HDD. To capture these characteristics, the flow is assumed laminar, incompressible and the computed result will show the steady-state condition as the flow parameters stabilize from the calculation. Snapshots of the velocity and pressure profiles in the HDD with arm fixed at the three different locations are shown in Figures 4.8-4.9. The colors are calibrated to the magnitudes of the velocity and static gauge pressures.

Figures 4.8(a)-(c) shows the computed airflow velocity vector plots for arm positioned at ID, MD & OD respectively. As the arm moves towards the OD, an obstruction to the flow is formed between the slider and disk boundary, and undergoes change of flow directions towards the outer radius therefore generating larger magnitude flow. Correspondingly, the pressure contour as illustrated by Figure 4.9(a)-(c) indicates an expansion of the higher-pressure zone towards the suspension as the arm is shifted towards the OD.

While most of the changes in velocity and pressure occur near the disk surface, the magnitudes of velocity and the static gauge pressure remain undisturbed in the regions outside the region bounded by the spinning disk perimeter. The corresponding pressure contours shown in Fig 4.9(c) indicates a smooth increment of total pressure (gauge) from the spindle motor wall towards the outer region. The pressure over the inner regions of the disk surface is almost symmetrical about the rotation axis while the slight distortion near the OD regions could be attributed to the presence of the arm. The highest static gauge pressure appears near the re-circulation filter location.

Figure 4.8: Snapshot of laminar field in the drive. Plot of Velocity magnitudes on plane 1 (a) Arm at ID (b) Arm at MD and (c) Arm at OD.

(a)

(b)

(c)

Figure 4.9: Snapshot of laminar field in the drive. Plot of static pressures on plane 1 (a) Arm at ID (b) Arm at MD and (c) Arm at OD.

The contours of the velocity magnitude and static pressure variation along the Z-axis are presented for both plane 3 and 4 in Figs 4.10(a)-(b), both cutting through the center axis of the motor-hub. The velocity reduces abruptly away from the disk surface indicates a weak flow induced by the single rotating disk. The highest velocity occurs at the edges of both planes and with an almost stagnation magnitude at the various regions away from the disk colored with dark blue in Fig 4.10(a). In Fig 4.10(b), the maximum static pressure occurs at the underside of the disk at the edge on plane 3 (highlighted in red). The high pressure at the edge of the disk creates a positive force that acted on fluid parcels towards the negative pressure zone, under the disk towards the motor-hub wall when effect of rotation is neglected. However, in the case of strong convection due to the disk rotation, centrifugal force is much significant than the negative pressure force on the fluid element and forces streamlines outwardly as the computation increases.

Figures 4.11(a) shows the velocity contour plot on plane 1 for the complete velocity range between 0 to 5.088 m/s. The contour plot illustrates a clearer view of a gradual decrease of velocity magnitude from edge of disk to the surrounding and a bigger region of back flow at the dedicated ramp location as compared to the vector plot of Figure 4.12(a).

Velocity shedding is also observed at the wake of the slider. A conspicuous reduction on the velocity magnitude is also seen near the suspension at either side (ID and OD) of the structure. This is accounted by a loss of momentum suffered by the oncoming flow when it is blocked by the arm suspension that crosses the rotating flow path. Further upstream of the arm also shows some flow dampening effect. This could be due to the pressure buildup at the suspension upstream as a result of the suspension wall resistance.

Figure 4.11(b) shows streamlines that are released from the upstream of the arm along plane 1. A total of 30 points at equal intervals are released along a linear path from ID to OD. These streamlines advance smoothly and remain laminar from the point of release to the near region of the suspension. The flow paths shifted outwardly in a fashion similar to the contour of the local pressure distribution as shown by Figure 4.9(a) at the upstream of the arm. This change of pressure is partially also affected by an

expansion of space at the perimeter of the breather filter. The expansion of space caused a slight drop in pressure and velocity magnitude (Figure 4.8(a) and 4.9(a)). This is true as the streamlines that are released at the ID, nearer the hub wall are unaffected and remain on a laminar path that follows a circular movement as seen in Figure 4.11(b).

Figure 4.12 shows a clearer snapshot of the solo streamlines without the disk for better clarity of presentation. The streamlines are tracked as they pass the suspension arm and re-entering under the breather filter. The air volume experienced an expansion in z-height space, flowing out of the breather filter at the upstream of arm and then constricting into a small space under the filter again after passing the arm. It is observed from the plot in Figure 4.12 that streamlines nearer the motor-hub favors an inward direction, completely opposing to the centrifugal force. These streamlines are drawn by the large inertia force driven by the rotating boundary wall of the motor-hub while avoiding the constriction in space under the breather filter.

It can be observed from the similar streamline plot of Figure 4.12, that the flow path along the outer edge of the disk shifted outwardly to avoid the constricted space under the filter and aided by an expansion of space outside the disk and centrifugal effect.

The assessment of velocity vectors along the Z direction can also be seen with Figure 4.13, showing a normal plane cutting the center of the releasing line. The colored vectors are calibrated to the velocity magnitude and movement through the arm can be observed from this plot. A gradual increase of speed is observed as air is drawn into the arm between the top/bottom suspensions as colored by the green vectors. From this plot, very little air volume flows over the arm due to the little space constraint by the top-cover and top arm surface. Air that is drawn downwards represented by the darkest blue vectors, expand into a larger space and goes into circulation away near the base thereby also results in a loss of momentum.

Figure 4.10: Contours plots of (a) Velocity magnitude (b) Static (gauge) pressure on plane 3 & 4.

(b)

Figure 4.11: Perspective view of arm at ID for (a) Velocity magnitude contour (b) Streamlines

Figure 4.12: Streamlines trace without disk

Figure 4.13: Velocity Magnitude Vectors before entering arm suspension

4.2 COMPUTED AIRFLOW CHARACTERISTICS IN A GEOMETRICALLY EXACT HDD ENCLOSURE

In this section, an improved version of the computational mesh that is developed from a geometrically exact HDD enclosure model is used. This exact model consists of a highly detailed arm and an exact replicate of the load/unload ramp. The enclosure wall contours are also closely replicated to the real HDD. The arm is rotated to three different positions (at the ID, MD and OD) and repeated in the computations for comparative studies with the basic model from earlier results. Also, the effect of a re-circulation filter is demonstrated by introducing a porous-jump 2D wall boundary. The results are compared and discussed.

The slider dimension is referenced to that of the industrial standard of a PICO [12]. A typical PICO slider measures 1.25mm in length, 1.0mm in width and 0.3mm thickness. For the ease of modeling, the thickness in this case is extruded towards the disk surface by 0.03mm to achieve the HDI of 5 μ m. This is the limitation the meshing software (*Gambit*) can achieve to mesh the HDD enclosure with tetrahedral elements in a single volume. The new HDI dimension brings closer the gap from the earlier 9 μ m with the basic model case. In either model, the introduction of a physical slider affects prominently on the flow distribution as can be seen from the velocity and pressure plots.

The slider is part of the arm component assembly connected via the dimple protruded from the suspension. Figure 2.1 shows an overview of the exact model that was described.

In this simulation, the flow parameters remain exactly unchanged with the previous study in Chapter 4.1. Snapshots of the velocity and pressure distribution in the HDD enclosure with the arm fixed at three different locations are shown in Figures 4.14-4.15. The airflow distribution as shown by the velocity vector plots on plane 1 in Figures 4.14(a)-(c) shows similar behavior to that of the basic model. The velocity vector plots show more randomness in the same plane 1. It is found that the interaction between the complex geometry and flow produces localized variation due to obstacles located at the various regions and especially within the rotating disk, like the arm suspension.

(a)

(b)

(c)

Figure 4.14: Snapshot of laminar field in the exact drive model. Plot of Velocity magnitudes vectors on plane 1 (a) Arm at ID (b) Arm at MD and (c) Arm at OD.

(a)

(b)

(c)

Figure 4.15: Snapshot of laminar field in the exact drive model. Plot of total pressure contour on plane 1 (a) Arm at ID (b) Arm at MD and (c) Arm at OD.

Figure 4.16: Static pressure (gauge) contour under the HDI plane

Figure 4.17: Velocity magnitude in vectors form illustrating the flow phenomenon around the slider

(a)

Figure 4.18: Pressure gradient across plane 1 with arm positioned in ID (a) $dP-dX$ (b) $dP-dY$.

Figure 4.19: Temperature Contours on plane 3 and 4

The slope part of the ramp blocks the rotating flow at the OD where angular velocity is highest due to a larger moment radius arm. The ramp attributes resistance to the flow causing air stream to flow downwards into the gap between the ramp and disk. The constriction in space also results in an increase of velocity magnitude in the localized zone. The velocity plots in Figures 4.14(a)-(c) also show shifting in the re-circulation of airflow at the upstream of the ramp with different arm positions. As the arm moves from the ID towards the OD, the intensity of the back flow at the upstream of the ramp becomes more pronounced. The magnitude of the back flow is enhanced by the position of the arm, especially at the OD where air streams with higher angular velocity flow along the arm axis, towards the ramp sidewall.

Figure 4.14(c) also shows a ramping up of velocity magnitude as the arm moves closer to the OD. The effect of a squeezed volume is observed as the flow passes between the suspensions and confined at the other side by the perimeter of disk and resisted by slope of the ramp.

In the same figures 4.14(a)-(c), an air stream colored in light blue is also noticeably flowing through the re-circulation filter holder. It is assumed complete permeability for air to flow through the filter and therefore it is omitted in the calculation. The red spots at the several regions at the OD also indicates higher velocity are results of an expansion in z-height space from the under side of the breather filter. Where the velocity magnitude under the HDI is very close to a stagnation value due to the thin gap height and upstream flow resistance from entering the zone. Figure 4.15(a)-(c) show the corresponding static pressure (gauge) contours.

Similar to the simple model, the higher total pressure zone lies towards the outer edge of disk, due to a higher angular velocity, therefore dynamic pressure is larger. It is observed that the expansion of the high pressure zone towards the OD as the arm moves outwardly is inconspicuous from the contour plots. This minimization of pressure variation could be partly due to the ramp, which stabilizes the flow by concentrating the fluctuation through channeling energy towards its gap between the ramp and disk spacing where the constriction creates high resistance to flow.

The maximum and minimum pressure occurs within the HDI location. Figure 4.16(a)-(c) illustrate the static pressure under the HDI corresponding to the three positions. It was found that the obstacles due to the slider block causes significant pressure gain at the trailing edge of slider. While at the leading edge, the expansion in space leads to a pressure loss. While the results may be a large estimate from the CFD simulation, it may provide parametric relevancy to the static simulation of air-bearing study within complex geometry. The inclusion of slider changes the flow parameters in its vicinity and is an important factor in particle trajectories analysis.

Figure 4.17 shows the vectors colored by velocity magnitude on the flow around the slider and an observable large velocity shedding occurs at the wake of the slider. Stagnant point is also noticeably seen at the wake of the slider block. Back flow is also observed when the vectors are being drawn into the stagnation zone behind the slider.

This observation correlates well to Lennenmann [2] in his experimental observation with a double spinning disk and beam inserted between the two disks. Although vortex shedding is not seen in his simulation that could be a result from the orientation of two close position disks rotating in the same direction, spot of vortices could however be seen in the other areas of thin gaps in the spinning disk region attributed by the spinning force which is seen between the ramp and disk.

Figure 4.18(a)-(b) plots the pressure derivative along the x and y axis of plane 1. The pressure gradient indicates a preference to streamlines along its direction and is useful in the prediction to particles trajectories.

Figure 4.19 shows the temperature contour on both plane 3 and 4 where a smooth transition of heat reduction is seen from the motor-hub wall to the MD zone. The temperature contour follows a parabolic profile under the disk space where a cavity exists, favorable for advection and convection to take place.

The breather filter that is adhered to the top cover above the disk attributes as an obstacle to the approaching circulating air stream. Air is diverted around and under the filter wall along the spinning disk direction. Figure 4.20(a) shows the velocity vectors plot taken across a plane normal to the end of the breather filter. Figure 4.20(b) shows a close up on vectors at the entering zone the breather filter. A very gradual diversion of

flow vectors is observed due to the step inclination. The amount of vectors angle change is a function of speed and inclination step angle too. Therefore, a round curvature at the edge of the obstacle should favor a more laminar flow and lower the possibility of air-ridden particle to impinge onto disk. However, it is also understood that an increase in the rotational speed would possibly increase impact angle driving air vectors downwards at a higher speed. This is detrimental to the reliability of HDD.

Figure 4.21 plots the drop in linear velocity along the Z-axis between the disk and the top cover. The velocity profile can be described by the classical Couette flow theory as the spacing is small and the air could therefore be viscous. The laminar stream flows between two parallel surfaces, the disk of which is moving relative to a static top cover. Therefore, viscous drag force exists in the boundary layer for both top and bottom of the spinning disk to act on the fluid drive the flow.

Figure 4.21 shows a steeper decline gradient at Part I. Velocity decreases at an average of 0.78125 m/s per 0.1 mm distance from the spinning disk. Part II shows a slight gradual decline at only 0.3629m/s per 0.1 mm. The drop in linear velocity is symmetrical for both sides of disk.

Figure 4.22 shows the tracking on streamlines released from the upstream of the arm at the OD position. A total of 30 points evenly spaced from ID to OD are tracked. Three distinct flow regions can be distinguished from this plot. (a) Turbulent wake region downstream of slider. (b) Re-laminarized region downstream of arm as the streams rejoin the laminar path. (c) Velocity shedding from slider. Repeated simulation are performed for all three positions and found to be highly repeatable.

Figures 4.23(a)-(b) show the computed velocity vector plots with vectors color representing the magnitude of velocity on the midsection of the arm axis on plane 5. The disk in between the sliders is omitted for visual clarity. Figure 4.23(a) shows high magnitude in flow speed near to the rotating disk and reduces in large magnitude upon slightly moving distance from the spinning surface. The vectors also show a re-flux at the leading edge of slider, moving upwards and at a small back angle upwardly.

At the outflow region shown by Figure 4.23(a), a darker blue arrow indicates low velocity and especially low flow at the wake of the slider. Figure 4.23(b) shows the

corresponding contours on the arm assembly with an overlay of the surface pressure with the velocity vectors cutting along a plane through the slider. The pressure at the slider surface near to disk indicates a higher value at the inner edges of the slider. The outflow region (indicated on this figure near the trailing edge of the slider) experiences a low pressure as a result of airflow around the slider.

Negative pressure induces force that favor streamlines towards it and since light particles ride on streamlines, one could use pressure force to predict the trajectories of particles to a good sense. A room with a lower pressure than its surroundings will cause air to flow into the room from the outside whenever a door or window is opened. In this aspect, there would be regions of low pressure by identifying the respective lower pressure zones, one could relate it with streamlines behavior.

Figure 4.24(a)-(b) shows the negative gauge pressure taken from plane 1 (between slider and top disk surface) and plane 2 (between slider and bottom disk surface). The negative pressure regions across plane 1 and 2 are symmetrically identical.

The negative zones near the rotor-hub wall as shown by Figure 4.24(a) are indications of possible zones for flows favor region, wherever the inertia force induced by rotating disk is weak. This is likely to be at a distance away from disk surfaces. Streamlines are highly random however certain trends could be predicted, depending on the release points. Figure 4.22 shows streamlines tracked from plane 1, along the radii from the upstream of arm and tracked for 0.37 seconds, equivalent to 22 revolutions of disk. Streamlines re-circulate at the upstream of ramp, outside the disk, at the negative pressure region before being drawn into the rotating zone.

The computed streamlines near at the downstream region of the slider as the arm is held stationary at the OD is illustrated in Figure 4.22(a). It can be seen that the streamlines approaching the slider have a tendency to move upwards or downwards corresponding to the top or bottom sliders and diverge around the actuator arm regions. In order to visualize these streamlines, the disk in between the sliders is not shown in Figure 4.23.

(a)

(b)

Figure 4.20: Velocity vectors plot on a (a) normal plane cutting through the edge of breather filter (b) close up view at the entry to breather filter.

Figure 4.21: Velocity profile from top disk surface to top cover, Couette flow

Figure 4.22: Streamlines tracked from the upstream of arm assembly for 500 steps

(a)

(b)

Figure 4.23 (a) Velocity vector plot along plane 5 (b) Slider contact (air bearing) surface pressure plot overlays with velocity plot on plane 5

(a)

(b)

Figure 4.24: Negative gauge pressure across (a) plane 1 (b) plane 2

Figure 4.25: Streamlines tracked from filter exit

(a)

(b)

Figure 4.26: Streamlines (a) around slider (b) from upstream of the slider

The interaction between the re-circulation filter and the airflow in the HDD enclosure is being simulated and discussed. The filter is modeled by a two-dimensional wall, seated diagonally across the filter holder as shown by Figure 4.27. The boundary condition for the filter is defined with a porous jump boundary. With this boundary type, the filter is modeled as a thin "membrane" that has known velocity (pressure-drop) characteristics. It is essentially a one-dimension simplification of the porous media model available for cell zones. This simplified model is used instead of a full porous media model because of its robustness and yields better convergence. The thin porous medium has a finite thickness over which the pressure change is defined as a combination of Darcy's Law and an additional inertial loss term as described in chapter 3.5.

Figure 4.28 shows the convergence history of the mass flow rate across the re-circulation filter, where x-axis plots the iterations to the computation. The air stream fluctuates as it passes through the filter membrane while the calculation continues to reach flow stability. The mass flow rate converged after about a 1100 iterations, and stabilizes at the absolute value of $1.46\text{e-}06$ kg/s. The filter has a surface area of $5.57\text{e-}06$ m² and extends across the filter slot. Figure 4.29(a)-(b) shows the velocity vector plots across plane 1 for the cases with filter and without filter respectively. The filter provides flow resistance and evolving a higher velocity fluctuation. The flow velocity at the OD of the disk also experience increase in magnitude due to the filter resistance & diverge of flow vectors. Similar location like the downstream of the arm and in the vicinity of ramp also sees the trend.

Figure 4.30 show an arbitrary line that passes through the filter through its center. Figure 4.31 shows the velocity magnitudes plotted along this line. The vertical axis indicates the linear velocity of flow while the horizontal axis plots the length along the line in the same orientation to Figure 4.31. A vertical line is drawn across the X-Y plot on Figure 4.31, indicates the intercepting position of the re-circulation filter. It is observed that the linear velocity at the beginning of the line, upstream of filter is higher for the case without filter. This is due to an unobstructed flow through the designated filter location therefore the increase in magnitude. While fluctuation within 0.1 m/s are

also observed along the line. Flow velocity starts to increase nearer to the rotating disk at the downstream of the designated filter location. The inertia force on any fluid parcels will motivate the surrounding volume to move along the circular path of disk at this location, therefore also increasing speed. Comparatively, the velocity at the upstream for the case with filter is lower than that without a filter. Flow accelerated as it moves towards the filter wall due to the permeable membrane. The constant flow mass through a smaller cross section area increases the flow velocity. The left part of both graphs show similarity increase in the same velocity magnitudes toward the rotating disk.

Figure 4.32(a)-(b) plot the pressure contour on plane 1 for both cases with filter and without filter. Pressure ranges between -2 to 4 Pa are selected for these contour plots that illustrate the major variations in the filter vicinity. It can be observed from Figure 4.32(a) that the filter changes the pressure balance across plane 1 and thus the HDD enclosure pressure. The pressure at the upstream of filter shows larger variation with higher magnitudes as seen from the color plots. There is also a built up pressure at the ramp slope and an enlargement of upstream pressure at the opposite corner to the filter. Figure 4.32(b) shows a more stable pressure distribution across plane 1. The pressure across the filter is also plotted for both cases as shown by Figure 4.33. The pressure drop is typically proportional to the velocity at the near wall to the filter. The pressure across a non-obstructed flow shows linear declining in magnitude.

Figure 4.27: Re-circulation filter cutting diagonally across filter holder

Figure 4.28: Convergence history of Mass Flow Rate on Re-circulation Filter

(a)

(b)

Figure 4.29: Velocity vectors plot on plane 1 (a) Re-circulation filter (b) No Re-circulation filter.

Figure 4.30: Arbitrary line created through the filter for plotting flow variables

Figure 4.31: X-Y plot for velocity magnitude along arbitrary line comparing between filter effect and without filter effect.

(a)

(b)

Figure 4.32: Static pressure (gauge) contour across plane 1 (a) Re-circulation filter (b) No Re-circulation filter, for pressure range between -2 to 4 Pa.

Figure 4.33: X-Y plot for pressure along arbitrary line comparing between filter effect and without filter effect.

The comparison between two cases of Laminar flow and turbulent flow computed with the Reynolds Stress Model (RSM) are presented. RSM is chosen as the pressure-strain terms approximate the process of energy redistribution that models well with rotating flows. Ali and Georgios [73] predicted the flow in a strong swirling vortex chamber using the RSM turbulence model and compared the computed and measured pressure drop across the chamber. The predictions provided by the numerical model shows clearly the forced and free vortex modes of the tangential velocity profile. The acquired results adequately agreed between the two tests.

Figure 4.34(a)-(b) show the velocity contours between the case of laminar flow and turbulence flow. Observations show the global flow distribution based on velocity & static pressure has no significant changes. The velocity magnitude at the close-up of ramp, as shown by Figure 4.34(b) shows slight increase in intensity. This could also suggest more erratic behavior at the ramp however effect is small and do not influence global distribution. Global velocities of the two cases therefore suggest identical between these cases. Figure 4.35(a)-(b) show some differences in static pressure range across the selected plane occurs at specific wake regions, slider, ramp, etc. The wake flow at downstream of slider re-organizes and eventually follow the laminar streamlines. The main difference in pressure distribution largely occurs under the slider, in the HDI region. The circled regions in the HDI shown in Figure 4.35(a) indicate low pressure, which is outside the plot range and not captured. As such, using laminar model to compute such low rotation flow is adequate for our prediction as the variation is insignificant to global distribution.

Figure 4.34: Velocity magnitude contour for (a) Laminar flow (b) RSM, Turbulence flow

Figure 4.35: Pressure contour for (a) Laminar flow (b) RSM, Turbulence flow

4.3 PARTICLE TRANSPORT IN THE GEOMETRICALLY EXACT HDD ENCLOSURE

The motivation for tracking particle trajectories in the HDD enclosure is to minimize particle-induced failures. Alumina (Al_2O_3) and Poly-Oxy-Methylene, (POM), representing ‘hard’ and ‘soft’ particles, were used for the following simulations. Using the converged numerical flow field inside the HDD, three groups of particles with varying sizes ($\phi 1.0\mu\text{m}$, $\phi 0.1\mu\text{m}$, and $\phi 0.08\mu\text{m}$) for both alumina and POM are released into the HDD enclosure with zero initial velocity at the various locations; near slider walls, ramp slope and along suspension. The particle trajectories are then tracked using the particle force balance model of Equation 3.16.

Alumina particles of various diameters are tracked by releasing them from the slider walls with the arm position at the MD. Alumina particles of diameter $\phi 0.1\mu\text{m}$ are released very close (i.e. $\approx 0.01\text{mm}$) from the four sides walls of the slider labeled as Wall 1-4 of both top/bottom sliders as shown in Fig 4.36 where the disk which lies between the top and bottom slider is kept invisible for ease of visualization. The locations of the particle release points are indicated by the red and yellow dots, evenly positioned. Walls 1 and 3 consist of 4 release points positioned at 0.25mm apart while walls 2 and 4 have 3 release points at 0.3mm apart.

Figure 4.36 Particles release from leeward of slider wall.

Thermal effects on the global air flow characteristics inside the 1-inch HDD are considered first by treating the spindle motor wall as one of the main heat sources (333K) inside the computational domain and the outer surfaces of the HDD enclosure as being subjected to a fixed temperature at 300K. The continuity, momentum, and energy equations are then solved and contours of velocity and pressure fields examined for the case of the disk spinning at 3600 rpm. It is found that while the thermal effects resulting from the specified heat sources do not alter the distribution of the pressure and velocity magnitude contours as shown by Figures 4.37(a)-(d), the temperature gradient in the HDD enclosure as shown in Figure 4.38(a)-(b) for plane 3 and 4, introduces thermophoretic force that affects the particle trajectories. The micron size particles suspended by the air rotation would experience a force in the direction opposite to that of the temperature gradient. Figure 4.39(a) plots the temperature derivation along its x-axis and Figure 4.39(b) plots the same function along its y-axis. $\vec{F} = (\nabla \cdot T)$

Significant temperature variations occur in the inner regions of the disk as compared to the outer regions of the disk. The force vector $\vec{F}$ acts along a negative $\nabla \cdot T$ direction as explained by Equation 3.22(a).

The computed particle trajectories released from slider wall as described by Figure 4.39 are shown in Figure 4.40 with colors representing local velocity magnitudes. The trajectories are computed using the DPM model in *Fluent*.

At the initial points the particles start out with low speeds and gradually reach higher speeds according to the ambient velocity field along the particle trajectories. The distance of the particle along the z-direction is tracked along the particle trajectory. The variation of this distance along the particle trajectory vs. the distance along the trajectory for four 0.1 μ m alumina particles released from the four particle release points on wall 1 of the top slider is shown in Figure 4.39. Particles are observed to move quickly towards the surface of the spinning disk acted on by the inertia force induced by the computed flow field and gradually fluctuate in the z-direction as they move along the trajectories. These fluctuations in height above the disk along the particle trajectories are tracked to provide insight on the possible impact of the particles with the disk surface. To minimize the error for computational limitation, the meshes near the disk surface are refined to a

maximum of 0.3 size intervals and with the near slider region with mesh close to 0.005 size intervals (corresponding to 5 microns). The face meshes on both top/bottom disk surfaces are set to grow in gradual increment of 1.08 from the boundary of the slider perimeter as shown by Fig 4.41. The highly refined surface meshes will subsequently produce a highly refined volume mesh at the near disk surface vicinity. This is important to the determination of micron size particles along the z-height variation. The disk surfaces are embedding type walls that will trap impinged particles onto wall irreversibly.

Figures 4.42-4.43 show the views on the x-y plane for the particle trajectories released from both the top and bottom sliders. Particles released from wall 2 of the top and bottom sliders are observed to travel along different paths. Instead of the particles being drawn to the proximity of disk surface, they appear to move through the top face of slider, away from the HDI to pass through the slider before joining the rest of the particles (Fig 4.44). Four alumina particles from the centre of each slider wall are selected to track their trajectories. Figures 4.44(a)-(b) shows the variation of the distance of the particles in the z-direction versus the distance measured along the trajectory from the release points for the four particles released from both the top and bottom sliders. Effects of gravity on the particle transport model were neglected for this case. Figures 4.45(a)-(b) illustrate the corresponding model which includes gravitational effect in the particle transport model. It appears from this comparison that gravity effects may not have much impact on the variation of the distance of the particles in the z-direction vs. the distance measured along the trajectory from the release points. These figures give insight into the possibilities of particle impact with the disk surface.

Figures 4.46(a)-(b) show the computed total (gauge) pressure contour and velocity vectors on plane 5 cutting through the middle line drawn along the length of the arm and parallel to the z-axis. The computed total pressure is the sum of static (gauge) pressure and the dynamic (gauge) pressure resulted from the airflow velocity. A higher pressure exists in the region marked as leading edge (flow entry) of the sliders and negative (gauge) pressure regions exist at the trailing edge of the sliders as indicated in Figure 4.46(a). Leading edge corresponds to Wall 2 of Fig 4.30 where particles path are

observed to fly away from slider surface. At this region, the negative pressure forces (Equation 3.21) dominates over both the centrifugal force (Equation 3.25) due to the rotating disk and the induced inertia force $\frac{d\bar{u}_p}{dt}$. The existence of this pressure differential along the Z direction for the leading and trailing edge explains the suction force that causes particles in the vicinity of the slider to be attracted towards or away from the disk surface.

Figure 4.47 shows the computed gauge pressure contours on plane 3 and plane cutting through the motor-hub. The presence of a cavity under the rotating disk as shown in Figures 4.47(a)-(b) produce a low pressure region and the pressure differential that exist between the cavity and its vicinity. This negative pressure zone draws particles away from the disk regions into the cavity regions. However, this effect is only significant to particle that is outside of the disk perimeter or distance greater than 0.15mm from disk surface as the induced inertia force could be weaker. As a result, particles are less likely to impact the inner regions of the disk during the particle transport. Particles released from the bottom surface of the disk are observed to be able to travel longer trajectories before impact compared to the trajectories of the particles above the spinning disk. This could also be due to the negative pressure gradient force acting downward in the negative Z-direction in the absence or insignificant of the other forces in that region.

Effects of thermophoretic forces on the particles trajectories are considered by repeating the simulation, neglecting the heat source at the rotor hub and neglecting the energy equation so that there will be no temperature gradient in the enclosure. Computed particle trajectories of alumina released from the top slider wall 1 corresponding to absence of temperature gradients in the enclosure are shown in Figure 4.48(a). Fig 4.48(b) shows the computed particle trajectories of alumina particles released from the top slider wall 1 in the presence of a temperature field created by prescribing a heat source at the rotor hub. Particles in temperature field experienced a thermophoretic force shows a more pronounced effect in addition to the centrifugal force of the rotating disk and heat gradient effect. This results in an average particle path moving away further from the rotorhub wall.

When the trajectories are tracked longer, one can see lesser revolution of particles around the spindle disk before been purged out fully by centrifugal effect. The effect of thermophectic forces is however only effective in the region nearer to the heat wall. Therefore, when particles are released from OD, less variation can be observed from the particle trajectories.

The ramp is made from a non-conductive polymer (POM), with highly smooth surface. However, the contact between the lift tab and ramp during load/unload of arm creates friction which then generates particles from surface. The ramp being a lower hardness material would therefore serves as a potential particle generator. Figure 4.49(a)-(b) shows the trajectories of particles released from the ramp slope. The particles used are Poly-Oxy-Methylene (POM), with density at 1.42gram/cc. This material has a low thermal conductive range between 1.25~1.7 kJ/kg°C.

The particles are spherical and have a diameter of 0.4 μ m. Figure 4.49(a) shows the trajectories of 6 POM released from the slope of the ramp tracked for 500 iterations. All particles are observed to travel on the outer edge of disk and deviated into the breather filter location. The different colors on the trajectories represent the velocity magnitude variation. It can be observed a reduction of velocity as the POM moves into the breather filter location (in blue). The change of vectors direction away from the tangential direction of the disk is due to an increase in velocity magnitude in the upstream of the breather filter and affected also by the shape of re-circulation filter contour (Not shown here) on the top cover. The centrifugal force or force in rotating reference frame of Equation 3.15 acted on the particles, pushing them outwardly, thereby suspending these particles at the perimeter edge of disk. At the same time, the induced momentum force from the spinning disk drives these particles along the rotating direction. Figure 4.49(b) also illustrates the behavior of the POM particles upon releasing from the slope. It is observed that the POM correlates well to a similar behavior to the alumina particles presented earlier. These particles are drawn quickly upon releases towards the spinning disk surface due to the flow-induced inertia force and without touching the disk, lifting away from the disk by a strong radial force due to the rotating frame into the main stream.

Figure 4.50(a)-(b) also shows the velocity magnitude plots and the corresponding static (gauge) pressure contours along plane 6, intersecting the mid width of ramp slope. Figure 4.50(a) indicates a higher velocity at the top gap of the ramp (in red) and a low magnitude flow outside the disk and around the ramp. The velocity magnitude at any given space is influenced by the spatial condition in the model and the high velocity may be due to a smaller height space at the top compared to the bottom volume. The static (gauge) pressure distribution on plane 6 is shown with Figure 4.50(b) and indicates a higher pressure at the underside of the ramp slope. The blue region shows a negative (gauge) pressure created by an expansion of volume due to the cavity space underneath the disk. This negative pressure creates a weak negative pressure force that draws particles towards the base plate. However, this force is insignificant to the inertia force induced by the spinning disk and as such, it will only effect a change to any parcels that is away ($> 0.15\text{mm}$, at 3600rpm) from the effective zone as described by Figure 4.12.

Figure 4.51 shows 2 display planes cutting the leading edge and the trailing edge of the re-circulation filter. These 2 planes are significant in the study of the re-circulation filter design in shaping up the flow as the filter covers a large area of the circulating disk surface. Figure 4.51(a) shows the full view of the velocity vectors plot along the leading edge of the filter with the arrows indicating the flow direction. The red vectors show a higher velocity towards the OD nearer the disk surface and relatively minute changes in the flow magnitude underside the filter at the top disk. Figure 4.51(b) shows the close up of the leading edge and also observed a downward flow vectors at the underside of disk due to air space expansion and the effect of negative pressure force.

Figure 4.52(a) shows the velocity magnitude vectors at the trailing edge of the re-circulation filter along plane 8 and Figure 4.52(b) shows the corresponding close up view on the edge. The velocity magnitude under the disk is higher than the top disk surface in plane 8 due to an elevated base plate toward the exit of the filter.

Figure 4.37: Total (gauge) pressure contours on plane 1 for case (a) with heat source (b) no heat source, plot between -10 to 10 Pa.

Figure 4.37: Velocity Magnitude vector plot on plane 1 for case (a) with heat source (b) no heat source

Figure 4.38: Temperature gradient at (a) (dT/dX) plane 3 and (b) (dT/dY) plane 4 cutting the motor axis.

Figure 4.39: Alumina trajectories in the vicinity of the HDI

Figure 4.40: Variation of the particle height in the z-direction vs. distance along the particle trajectory for particles released from wall 1 of top slider

Figure 4.41: Highly refined surface meshes on disk with a size growth of 1.08 from slider edges

Figure 4.42 - Alumina particles trajectories released from top slider

Figure 4.43 - Alumina particles trajectories released from bottom slider

Figure 4.44: Trajectories of particles released from the four side-walls of (a) top slider (b) bottom slider neglecting effects of gravity on particle transport.

(a)

(b)

Figure 4.45: Trajectories of particles released from the four side walls of (a) top slider (b) bottom slider including effects of gravity on particle transport.

(a)

(b)

Figure 4.46 (a) Total (gauge) pressure contours, and (b) Velocity vectors on plane 5 cutting through the middle line drawn along the length of the arm and parallel to the z-axis

(a)

(b)

Figure 4.47: Computed gauge pressure contours (a) on z-y plane 3 and (b) on z-x plane 4, cutting through the rotor hub.

(a)

(b)

Figure 4.48 (a) Alumina path lines releasing from the top slider without heat force (b) Alumina path lines for top slider with heat effect

(a)

(b)

Figure 4.49 – Particle Trajectories of POM (a) around the disk (b) released from the ramp slope

Figure 4.50 (a) Velocity vectors along plane 6 (b) Static (gauge) pressure along plane 6

Figure 4.51: Display planes showing the leading edge (plane 7) and the trailing edge (plane 8) of the re-circulation filter.

(a)

(b)

Figure 4.52: Velocity magnitude vectors (a) on plane 7 (b) close up at the leading edge of re-circulation filter.

(a)

(b)

Figure 4.53: Velocity magnitude vectors (a) on plane 8 (b) close up at the trailing edge of re-circulation filter.

4.4 FINE PARTICLE MODEL (FPM) ON PARTICLES INTERACTION

In aerosol dynamics, the particles are described by their particle size distribution, $n(m_p, \vec{x}, t)$ where m_p represents particle mass, $\vec{x}$ is the position vector and t is the time. For the current study, these particles are assumed to be spherical in shape and the sizes of particles released from the source wall has a mean diameter of $0.1\mu\text{m}$. Hesketh (1986) divided fine particles into four particle size regimes, namely, the free molecular, transition, slip flow, and continuum regimes, based on the particle Knudsen number, $Kn = \frac{2\lambda}{d_p}$ where λ is the mean free path of air and d_p is the particle diameter.

This section presents an alternative method of tracking particle movement as opposed to the Lagrangian method as discussed in Section 4.3. The FPM model here looks study the closely suspended particles in the enclosure and the effects of particles diffusion, external heat force and particle-particle interaction based on a coagulation kernel function is addressed. The underlying equations for this section are discussed in Section 3.4.

Alumina particles in air behave differently from the air in which they are suspended and also differently among themselves depending on their size, shape and composition. In order to study particle transport phenomena inside the HDD, alumina particles with diameter of 0.1 microns are used. These particles are released from the arm slider wall as indicated by Figure 4.54(a) based on a lognormal distribution. The surfaces other than the suspensions are assumed to be perfect sinks for this simulation, i.e. the particles are absorbed upon contact with the surfaces. Figure 4.54(b) shows the arm suspensions where particles are released.

The disk is then spun to 3600 rpm and the airflow field is solved with the transient equation assuming laminar flow. The computation finally converged to the steady state condition with all components stationary and the arm positioned in the ID.

In this study, three cases are considered in the attempt to understand the effects of the various forces such as heat and coagulation where only Brownian diffusion is considered. The first case assumes diffusion as the only effect while the second case

includes the thermophoretic force resulted from a heating source wall as indicated by Figure 4.4. The source wall extends along the full depth as a constant cross section cylinder in the HDD and is set as at 333K, while the other boundary walls are set at 300K. The third case considers the effects of particles-particles interaction with particles coagulation in the laminar case. Figure 4.44 shows the plot of percentage of cells in the 1-inch HDD with varying particle diameters. It is observed that the variation of the concentration of similar diameter particles lies particular on those falling between 0.03 and 0.09 micron. The 2 peak points in Figure 4.55 show where the 2 most representative geometric diameter lies in the computational domain. The heat force introduces energy to the enclosure particles thereby causing more erratic motion, which can be treated as a mild turbulence, small however significant enough to cause an effect in the 1-inch length scale HDD. It also suggests that the combination of the heat force with coagulation effect enhances the percentile of the highly populated particles diameter.

Figure 4.55(a)-(b) also illustrates the effect of heat on the particles concentration where Figure 4.55(a) shows the results of only Brownian diffusion process acting on the particles and Figure 4.55(b) shows the results based on the Brownian diffusion and the thermophoretic forces. Heat force pushes particles away in the direction of reducing temperature gradient thereby resulting in a particle free region as indicated by the light blue contour in Figure 4.55(b). Figure 4.56 traces the iso-surface contours of the particle concentrations based on 0.048 and 0.0886 micron referenced to the first and second peaks in Figure 4.54. On the same figure, a close up view on the source wall slider is shown. In Figure 4.55(a), the 0.048 microns size particle is diffused along circumference of the media disk following the airflow induced by the spinning disk. In Figure 4.55(b), the 0.0886 microns size particle is observed to be more resistant to the influence of Brownian diffusion as it “clings” to the vicinity of the slider wall.

With the aforementioned results, the limitation to exploring the problem further lies greatly on the computational power and achieving a sophisticated and good quality mesh, capable of moving the arm as in the simple HDD geometry presented in Figure 1.

(a)

(b)

Figure 4.54. (a) Wire frame top view of an 1-inch HDD (b) Arm suspension with slider as releasing source

Figure 4.55. Percentage of cells in the HDD enclosure populated with particles of given diameters at steady state.

(a)

(b)

Figure 4.56. Particle concentration distribution due to effect of (a) diffusion (b) diffusion with heat force

(a)

(b)

Figure 4.57 - The evolution of particle concentration on an iso-surface of particle diameter, (a) 0.048 micron, and (b) 0.0886 micron. Contours represent the particle concentrations.

4.5 LOCAL MESH REFINEMENT USING SOLUTION-ADAPTIVE METHOD

Some limitations on the grids exist when higher resolutions and accuracy to the solutions are required in selected regions. To enable improvement on mesh in the course of analysis, solution-adaptive refinement is adopted. This method enable cells to be added where needed, thus enabling the features of the flow field to be better resolved. When adaption is used properly, the resulting mesh is optimal for the flow solution because the solution is used to determine where more cells need to be added. Thus, computational resources are not wasted by the inclusion of unnecessary cells, as it occurs in the structured grid approach. Also, the effect of mesh refinement on the solution can be studied without completely regenerating the mesh.

Figure 4.58(a) shows the velocity magnitude contour at the ramp vicinity with arm positioned to the OD. With the adapted mesh as shown in Figure 4.58(b), the concentration of a higher velocity colored in red is observed prominently at the downstream of the slider toward the disk edge. A larger range of velocity variation is also captured as indicated by the range scales of both plots.

Mesh adaption is performed at the ramp region where high velocity variation is observed as the arm moves towards the OD. Two different planes on the grid distributions above 2 elevations (40 μm and 90 μm above top disk surface) from the disk are illustrated in Figure 4.59(a)-(b) from the solution-adaption result. Figure 4.59(a) shows the grid distribution at 40 μm above disk. Refinement is seen nearer the slider wall due to a change in velocity magnitude. However, at 90 μm level, the flow experienced less magnitude of variation across the x-y plane, therefore the adaption mesh shows little refinement in Figure 4.59(b). The solutions are then further iterated to achieve new results for the adapted mesh problem. The computed velocity contours are compared between the 2 cases. Figure 4.60-4.61 shows the computed velocity contours between the 2 cases on 2 different planes. Figure 4.62 shows a more distinct region of the flow at the ramp region.

Vorticity is the measure of the curl of velocity vectors and often gives a good indication to the generation of a localized turbulence. With the mesh being adapted at certain region, it is possible to retrieve valuable information with quantitative datas. The vortical region around the slider can be seen from Figure 4.63.

A comparison of the vortices variation contour plot at the near wall of the slider is taken at $10\text{ }\mu\text{m}$ above the top disk surface for analysis. Figure 4.56(a) shows a slight change in the color contour at the leading/trailing edge of slider. Figure 4.56(b) shows the close up view of the vortical region due to a refinement of mesh.

Figure 4.57(a)-(d) shows the close-up on slider from the top view. Figure 4.57(b) has the edge a higher vorticity magnitude cells and this could be due to the on-coming flow directly in contact with the edge. Figure 4.57(d) shows the edge that is blocked from direct flow, and the kinetic energy is dissolved during diversion to result in a minimum intensity.

The position of the arm relative to the ramp is as shown by Figure 4.58(a). The effect of mesh refinement at the region of high velocity enables the identification of a larger region of high vortices surrounding the slider, extending to the lift tab, which is nearer to the ramp.

It appears that the solution-adaptive method yielded useful information at the vicinity of the slider, further enhancing the resolution of the flow, which attributes to a better understanding to the localized turbulence. Solution-adaptive grid refinement can be used to increase and/or reduce grid density based on the evolving flow field, thus providing the potential for more economical use of grid points (and hence reduced time and resource requirements). The computed solution is used as the boundary conditions for the structure studies. Likewise, the data file could be incorporated into ANSYS for Fluid-Structure-Interaction computing.

Figure 4.58 – Velocity Contour with (a) No adapted mesh (b) adapted mesh.

Figure 4.59 – Grids distribution at (a) 40 μm (b) 90 μm above disk

Figure 4.60 – Velocity contours at (a) 40 μm (b) 90 μm above disk before mesh adaption

Figure 4.61 – Velocity contours at (a) 40 μm (b) 90 μm above disk with mesh adaption.

Figure 4.62 – Vorticity Magnitude contour taken from plane at 10μm above disk (a) Before adaptive mesh (b) With adaptive mesh.

Figure 4.63 – Vorticity Magnitude at cells zone (a) Left Top (b) right top (c) left bottom (d) right bottom

Figure 4.64 (a) Arm & Ramp assembly (a) Arm superimpose with vorticity magnitude above 40,000 (1/s).

4.6 FLOW IMPROVEMENT WITH WALL SHAPING

Flying particles within the disk perimeter are hazardous to HDD operations as particles may impinge onto surfaces in finite operation time. The air space above the disk is shaped largely by the existing breather filter adhered to top cover. From the flow distribution, air entering the filter has tendency to drive streamlines downwards as shown by vector on Figure 4.23. To smoothen the transition flow due to a 90 deg step, an air-guide could be incorporated that would reduce a large flow angle change and to improve outward purging of possible flying particles.

Two designs are suggested. Figure 4.65 shows an air-guide incorporated into the breather filter to guide flow outward to the OD, thereby minimized flying particles impingement. The air-guide is molded to the edge of the breather filter away from the arm. Wake flow at downstream of arm is more erratic and the guide will smoothen the flow (laminar) by diverting outwards. The contour of air-guide is angled along the rotating flow direction to avoid large direction change. Figure 4.66(a)-(b) also shows the top/bottom view of the air-guide.

Figure 4.67 shows another alternative design to flow improvement targeted at flow under the disk space. An isolated air-guide could be designed and placed under the disk to guide flow at the bottom disk outward. Figure 4.68 shows the assembly view of the guide at the bottom. Due to the additional z-height space available, it is suggested that a taper air-guide is used for the bottom air space. Figure 4.68(a) shows the front end (ID) of air-guide with smaller gap, as the pressure is lower in the ID. Figure 4.68(b) shows the rear end of air-guide with larger gap space, so that relatively higher pressure at OD is better released outside disk perimeter.

Figure 4.69 shows the particle trajectories of released Alumina of 0.1 μ m and interacting with the air-guide. It shows particles instead of flying around below the disk surface in its many revolutions now are guided outside the spinning disk perimeter region. With optimization on the shape, particles flying inwards due to lower pressure region could be minimized.

Figure 4.65: An air-guide incorporated onto breather filter (homogenous material) to guide streamlines to the outer-disk and minimize impingement of particles onto disk surface.

Figure 4.66: Top/bottom view of air-guide with breather filter.

Figure 4.67: Air- at bottom disk to guide particles away to outer disk

(a)

(b)

Figure 4.68(a): Front end (ID) of air-guide with taper groove to capture low pressure flow. (b) Rear end (OD) of air-guide with enlarged opening to ease flow purging at higher pressure.

Figure 4.69: Particle trajectories of 0.1μm spherical alumina with bottom air-guide. Alumina particles are guided outwardly.

CHAPTER 5

CONCLUSION AND RECOMMENDATION FOR FURTHER WORK

5.1 CONCLUSION

Numerical simulation using the complex geometry of the Hard Disk Drive (HDD) enclosure has not been extensively explored in the past due to the limitation with computer speeds and a robust meshing tool. In this work, a highly complex model of an 1-inch HDD enclosure model is created using unstructured mesh elements. The head-disk interface (HDI) as a result of the clearance between the slider and the rotating disk is also incorporated in the domain by a constant gap spacing of 5 microns. The co-existence of the computational model of the HDD and the 5 microns HDI within a single entity volume poses technical challenge in the creation of the geometry and the subsequent meshing process. Kirpekar and Bogy [74] simulated the airflow field in the Hard Disk Drive (HDD) using a simplified geometrical model. They reported on large eddy simulations of the turbulent flow in the HDD using a commercial CFD code on this simplified model. As such the simplified model presents limited information. The current model used in this work is a complete geometrical model as defined by Seagate. No simplifications on geometrical artefacts were made in this work.

Using the complex HDD model, the computed results provide more realistic flow analysis, especially in capturing the flow structures in the vicinity of HDD components and artefacts and also near the HDD walls. The computed particle trajectories can also be simulated and visualized using the computed flow results, thereby providing in-depth understanding of particle transport around the HDD artefacts and also in the design of particle filters and their optimal placement within the HDD. Potentially, this model could also be used for various case studies. For example, turbulent simulations relative to higher disk spinning speeds can be further investigated with such computational model by just increasing spinning speeds. The complex 1-inch HDD enclosure model also accounts for the effect of the pressure drop across re-circulation filter. The re-circulation filter is modeled and taken as a porous-jump boundary wall, based on the Darcy's law. The CFD model serves as an important tool for the optimization study pertaining to filter

placement within the HDD enclosures. The simulation also provides valuable insights for particle trapping with various filter orientations.

The tracking of particles in the HDD enclosure is calculated based on the Euler-Lagrangian and Eulerian approaches. In both cases, the injected particles volume are less than ten percent of the flow volume and which is consider insignificantly to result in changes to the continuous flow field. It was also observed that while thermophoretic force released from the wall of the rotor-hub affected particle trajectories, soft polymeric material with low thermal conductivity however is exception by such condition. From the computed result, it was also observed that gravitational force has insignificant influence to the micron sizes particle trajectories.

The Eulerian model of the FPM that considers the interaction between particles collision shows good agreement between aerosol dynamics in describing particle-to-particle interactions within the HDD enclosure. The results from the FPM also observed that larger mass particles favors flow stagnant region towards the corners especially where the magnet is located. Temperature gradient is formed from the heat source wall producing a thermophoresis force that drives particles along the direction of decreasing temperature gradient, aiding to the centrifugal force. It was observed that thermophoresis force has influence in causing change to thermal conductive particles. One possible explanation could be due to molecular excitation at the near heat wall region thereby increasing chances of collision with neighboring particles to form larger mass particles. The result of particles-particles interaction (coagulation) forms larger mass particles, and increase the ease of particle transport due to larger surface area and therefore higher induced inertia forces on these particles from the rotating flow.

The final results appear that the computational study on the airflow characteristics and particle transport modeling in the 1-inch HDD has yielded useful predictions that may be useful in guiding the design and placement of filtration systems (re-circulation filters, etc) within the HDD enclosure.

Some observations are also captured from the computed results, on the various distinct flow characteristics inside a HDD enclosure when operated differently. The general flow velocity increases as the arm moves from ID to OD. The protrusion of the ramp structure into the rotating disk region attributed flow magnitude changes at the

outer disk and also generating vertical structure that increases localized turbulence. The increase in vortices strength indicates possible flow-induced modulation of the HGA at that region. While the arm moves away towards the ID, the restriction to flow is less prominent due to space expansion radial wise.

The velocity magnitudes taken along the normal distance from disk surfaces towards the enclosure for both top/bottom indicates Couette flow validation. Also, the pressure distribution along the leading and trailing edge of the slider displayed skewed parabolic contour.

We note that this analysis is not complete in some respects: an accurate representation of the boundary layer flow is not captured due to coarse mesh.

The results in section 4 which assumed a largely laminar flow is validated by repeating the computation with the Reynolds Stress Model. However, it would be meaningful if the test can be repeated by computing using the direct numerical simulation (DNS). However ideal, DNS may remain prohibitively expensive for several years to come. Secondly, actual measurement of the flow with various visualization techniques (LDV, PIV, etc) could be used as experimental validation on a selected case.

Lastly, the HDI allows slippage of a small laminar air stream as seen by streamlines. It was also noted that whilst the HDI is included in this research, it is based on a large prediction to the nanometer order. The results from these simulations provide the design group in the Research Center to validate their design assumption without committing in large cost to fabricate the mould for enclosure casting or molding. The success of this work leads to the study of highly complicated geometry with low uncertainty of failure in meshing. The discrete phase modeling is also recommended as the tool for failure analysis, drive reliability and particle robustness test. CFD technique therefore provides the HDD designers a cost effective method in flow related analysis to produce a particle-free HDD.

5.2 RECOMMENDATION FOR FUTURE WORK

In addition to this work, some recommendations for further work are suggested. The computational mesh could be explored with the various turbulence models (LES, DNS, etc.). The boundary layer effects can be included by constructing finer mesh, which satisfied the boundary condition to understand the influence of the flow at the near disk region thereby accurately predicting impingement of particles on walls.

The airflow result can be enhanced with the coupling to the Finite Element Method (FEM) where the forces are exchanged at the boundary between the fluid and structure using ANSYS one-way coupling model. Fluid-structure interaction (FSI) can be explored to understand the impact of the flow on the arm modulation during read/write. In addition, the mis-track registration of the slider could be modeled.

A user-defined function (UDF) code could be written to move the arm dynamically. The transient simulation 3D animation could be played back so that the transient nature of drive operation can be well translated to the team.

The pressure or velocity profile at the leading/trailing edge of the slider can be computed with the current CFD method. The global flow effect due to geometry influence is therefore addressed to provide the boundary conditions to the HDI. The acquisition of this data can be used as the boundary conditions to a molecular dynamic codes which solve for the HDI where the continuum model no longer valid due to a low Knudsen number. The exchanging of information between these 2 codes would be an intensive subject worth researching into.

In the FPM computation, other forces like electromagnetic forces should be modeled to ensure more realistic prediction for the particle transport. The coagulation process can also be improved to include other forces such as gravity. It is hoped that through these improvements, the influence of the various forces could be captured for a given concentration of particles.

The aerosol model should be applied with other types of particles such as the outgas produce by the spindle motor lubricants that is suspected to evaporate into the enclosure with prolong heating. The result provides useful information to the design center in outgas control. Lastly, corresponding experimental should be done to validate the numerical simulation results and concurrently also improve on the modeling.

REFERENCES

- [1] A. Saito and H. Nishida, "Trends in HDDs for consumer electronics usage and their supporting technologies," in *Hitachi Review*, vol.56, no.1, 2007
- [2] E.Lennemann, "Aerodynamic Aspects of Disk Files", in *IBM J. Res. Develop*, pp.480-488, Nov 1974.
- [3] R. Kaneko, S. Oguchi, and K. Hoshiya, "Hydrodynamic characteristics in disk packs for magnetic storage", in *Review of the Electrical Communication Laboratories, Nippon Telegraph and Telephone Public Corp.*, Japan, 1977, vol.2, pp. 1325-1336.
- [4] S. D. Abrahamson, J. Eaton, and D. J. Koga, "The flow between shrouded co-rotating disks", in *Phys. Fluids*, vol. 1, no. 2, pp. 241-251, 1989.
- [5] J. Girard, S. D. Abrahamson, and K. Uznanski, "The effect of rotary arms on co-rotating disk flow," in *ASME Journal of Fluids Eng.*, vol.117, pp. 259-272, 1995.
- [6] A. Z. Szeri, F. Labbe, S. J. Schneider, and H. N. Kaufman, "Flow between rotating disks. Pt.1: basic flow", in *Journal of Fluid Mechanics*, vol.134, pp.103-131, Sept 1983.
- [7] D. Djistrka, G. J. F. Van Heijst, "The flow between two finite rotating disks enclosed by a cylinder", in *Journal of Fluid Mechanics*, vol.128, pp.123-154, 1983.
- [8] M. Zhou and C. P. Garner, "Particle image velocimetry measurements of the flow field within an enclosed rotating disc-stator system and comparisons with computational fluid dynamics results", in *Optical Diagnostic in Engineering*, vol.1, pp.9-21, 1996.
- [9] M. P. Escudier, "Observations of the flow produced in a cylindrical container by a rotating endwall", in *Journal of Exp in Fluids*, vol.2, pp.189-196, No.4, Dec 1984.
- [10] S. T. Thoroddsen, and J. M. Bauer, "Qualitative Flow Visualisation using Colored Lights and Reflective Flakes," in *Phys Fluid*, vol.11, no.7, pp.1702-1704, July 1999.
- [11] M. Tirumkudulu, A. Tripathi, and A. Acrivos, "Particle segregation in monodisperse sheared suspensions," in *Phys. Fluids* 11, 507, 1999
- [12] E. L. Koschmieder, *Bernard Cells and Taylor Vortices* (Cambridge University Press, Cambridge, 1993).
- [13] M. Roy and J. L. Brand, "Soft particle-induced magnetic erasure without physical damage to the media," in *Journal of Tribology*, pg.729-734, vol. 129, Oct 2007

- [14] K. Willeke and P.A. Baron, "Principles, techniques, and applications," in *Aerosol Measurement*, John Wiley and Sons Inc., NY, USA, pp.414-416.
- [15] D. Y. Lee, J. Hwang, and G. N. Bae, "Effect of disk rotational speed on contamination particles generated in a hard disk drive," in *Microsystem Technologies* 10, Springer-Verlag, pp.103-108, 2004.
- [16] H. S. Park, Y. C. Yoo, G. N. Bae, and J. H. Hwang, "Investigation of Particle Generation In a Hard Disk Drive During the Start/Stop Period," in *IEEE Transactions on Magnetism*, vol.35, no.5, Sept 1999.
- [17] C. J. Chang, J. A. C. Humphrey, and R. Grief, "Calculation of turbulent convection between co-rotating disks in axisymmetric enclosures", in *Int. Journal of Heat Mass Transfer*, vol. 33, no. 12, pp. 2701-2720, 1990.
- [18] J. A. C. Humphrey, C. A. Schuler, and D. R. Webster, "Unsteady laminar flow between a pair of disks corotating in a fixed cylindrical enclosure", in *Phys. Fluids*, vol.7, no. 6, pp. 1225-1240, 1995.
- [19] H. Suzuki and J. A. C. Humphrey, "Flow past large obstructions between corotating disk in fixed cylindrical enclosures", in *ASME Journal of Fluids Eng.*, vol.119, pp. 499-505, 1997.
- [20] I. Iglesias and J. A. C. Humphrey, "Two and three-dimensional laminar flows between disks co-rotating in a fixed cylindrical enclosure," in *Int. Journal of Numer. Methods Fluids*, vol.26, pp. 581-603, 1998.
- [21] H. Kazemi, "Mathematical modeling of flow-induced vibrations of suspension-head units in hard disks drives", in *Ph.D. dissertation*, Univ. Virginia, Charlottesville, 2004.
- [22] Z. X. Yuan, C. J. Luo, and X. Yan, "Airflow pressure and shear forces on a rotating, deformed disk in an open shroud," in *Comm in Nonlinear Sci. and Numer. Simulation*, vol.9, pp.481-497, 2004.
- [23] S. Ali and M. Damodaran, "Particle contamination in turbulent flow within a Hard Disk Drive enclosure", in *Progress in Computational Fluid Dynamics*, Vol. 8, No. 6, 2008.
- [24] Wu, S. C. and Chen, Y. M. (2002), "Phase-averaged method applied to periodic flow between shrouded co-rotating disks", in *International Journal of Rotating Machinery*, Vol. 8, No. 6, pp.413-421

- [25] Tzeng, H.M., Munce, A.C. and Crawforth, L. (1994) 'Quantitative airflow visualization between shrouded co-rotating disks', in *Transactions of ASME, Journal of Fluids Engineering*, Vol. 116, pp.494-498.
- [26] S. Ali, M. Damodaran, and Q. Y. Ng, "Computational Modeling of Particle Flow and Interaction in a Simple Hard Disk Enclosure with moving arms," in *Journal of Communications in Nonlinear Science and Numerical Simulation*, vol.9., pp.481-497, 2004
- [27] Y. K. Ng and Z. Y. Liu, "Prediction of unobstructed flow for Co-rotating multi disk drives in an enclosure", in *Int. J. Numer. Methods Fluids*, vol.35, pp.519-531, 2001.
- [28] H. Shimizu, M. Tokuyama, S. Imai, S. Nakamura, and K. Sakai, "Study of aerodynamic characteristics in hard disk drives by numerical simulation", in *IEEE Tran. Magn.*, vol.37, no.2, pp. 831-836, Mar. 2001.
- [29] N. Tsuda, H. Kobutera, M. Tatewaki, S. Noda, M. Hashiguchi, and T. Maruyama, "Unsteady analysis and experimental verification of the aerodynamic vibration mechanism of HDD arms", in *IEEE Trans. Magn.*, vol.39, no.2, pp.819-825, Mar, 2003.
- [30] M. Tatewaki, N. Tsuda, and T. Maruyama, " A numerical simulation of unsteady flows in HDDs", in *FUJITSU Set. Tech Journal*, vol.37, no.2, pp.227-235.
- [31] H. Song, M. Damodaran, and Q. Y. Ng, "Simulation of flow field and particle trajectories in hard disk drive enclosures", in *High Performance Computation for Engineered Systems (HPCES)*, Jan, 2004.
- [32] H. J. Goh, M. Damodaran, Q.Y. Ng, "Modeling Airflow and Particle Trajectories Near The Head/Disk Interface Region of a Small Form Factor Hard Disk Drive Enclosure," presented on *World Tribology Congress III, WTC2005-63633, 11~16 September, 2005, Washington DC, USA*.
- [33] A. Burgdorfer, "The influence of the molecular mean free path on the performance of hydrodynamics gas lubricated bearings," in *ASME J. Basic Eng.*, vol.81, pp. 94, 1959.
- [34] W. Huang, D. B. Bogy, and A. L. Garcia "Three-dimensional Monte Carlo method for slider air bearings," in *Phys. Fluids*, vol. 9, no. 6, June 1997
- [35] F. J. Alexander, A. L. Garcia and B. J. Alder, "Direct Simulation of Monte Carlo for Thin-Film Bearings", in *Physics of Fluids*, vol.6 (12), Dec 1994.
- [36] S. Zhang, and D. B. Bogy, "Motion of Particles in a Slider/Disk Interface Including Lift Force and Wall Effect", in *IEEE Trans. Magn.*, vol.33, no.5, 3166-3168, Sept. 1997.

- [37] C. J. Tsai, Y. H. Pui, and Y. H. Liu, "Particle detachment from disk surfaces of computer disk drives," in *J. Aerosol Sci.*, vol.22, no. 6, pp.737-746, 1991.
- [38] S. C. Garrick, K. E. J. Lehtinen, M. R. Zachariah, "Nanoparticle coagulation via a Navier-Stokes/nodal methodology: Evolution of the particle field," in *J. Aerosol Science.*, vol. 37, pp. 555-576, 2006.
- [39] S. Ali, M. Damodaran, and Q. Y. Ng, "Computational Models for Predicting Airflow Induced Particle Contamination in Hard Disk Drive Enclosures", presented on *AIAA Computational Fluid Dynamics Conference*, 6~9 June, 2005, Toronto, Canada
- [40] D. B. Bogy, J. E. Fromm, and F. E. Talke, "Exit Region Central Source Flow Between Finite Closely Spaced Parallel Co-Rotating Disks," in *American Institute of Physics, The Physics of Fluid*, vol.20, no.2, Feb 1977
- [41] John Benzi and M. Damodaran, "Parallel Direct Simulation Monte-Carlo Model for Simulation of Airflow in the Hard Disk Drive Slider Head Disk Interface Gap" presented at *Parallel Computational Fluid Dynamics 2007 Conference*, PARCFD'07, Antalya, Turkey, 21-24 May 2007.
- [42] John Benzi and M. Damodaran, "Computation of Head-Disk Interface Gap Micro Flowfields using DSMC and Continuum-Atomistic Hybrid Methods," Under review for publication in *International Journal of Numerical Methods in Fluids*, August 2008.
- [43] *Rhinoceros Software User Guide*, 2003
- [44] *Gambit*, version 2.2.14, User Guide, 2007.
- [45] J. Fowler, "STEP for Data Management, Exchange and Sharing", in British Library cataloguing-in-publication data, 1995
- [46] *Fluent* version 6.3.16, Flow Simulation Software User Guide, 2007.
- [47] *EnSight User Manual for Version 8.0*, June 2006
- [48] L. J. Piscitelle, N. Roberts and R. Segars, "Analysis and comparison of two mathematical models using effluent vapor breakthrough data from carbon filter beds", in *Carbon*, vol.36, nos.1-2, pp.99-104, 1998.
- [49] N. R. Amundson and Bohart, "A note on the mathematics of absorption in beds", in *J. Phys. Colloid Chem.*, vol.52, pp.1153, 1948

- [50] I. Nir, Y. Suzin and D. Kaplan, "The effect of airflow pattern on filter breakthrough in physical absorption", in *Carbon*, vol. 40, pp.2437-2445, 2002.
- [51] Disk drive filters, "Adsorbent recirculation filter," in *W.L Gore User guide*, 2006
- [52] R. Kerr, "A method for controlling skew on linked surfaces," in *Proceedings of 8th International Meshing Roundtable*, pp.99-2288, Sandia National Laboratories, Oct 1999.
- [53] W. Huang, D. B. Bogy, and A. L. Garcia, "Three-Dimensional Direct Simulation Monte Carlo Method for Slider Air Bearings," in *CML Research Reports*, Univ. of California, Berkeley, Jun, 1996
- [54] S. Fukui and R. Kaneko, "Analysis of Ultra-Thin Gas Film Lubrication Based on Linearized Boltzmann Equation," in *ASME Journal of Tribology*, vol.110, pp.253, 1988
- [55] G. K. Batchelor, "An introduction to Fluid Dynamics," in *Cambridge Univ. Press*, Cambridge, England, 1967.
- [56] T. J. Barth and D. Jespersen, "The design and application of upwind schemes on Unstructured meshes," in *Technical Report AIAA*, pp.89-366, 27th Aerospace Sciences Meeting, Reno, Nevada, 1989.
- [57] D. G. Holmes and S. D. Connell, "Solution of the 2D Navier-Stokes Equations on Unstructured Adaptive Grids," in *9th Computational Fluid Dynamics Conference AIAA*, June, 1989.
- [58] R. D. Rauch, J. T. Batira, and N. T. Y. Yang, "Spatial Adaption Procedures on Unstructured Meshes for Accurate Unsteady Aerodynamic Flow Computations," in *Technical Report AIAA*, pp.91-1106, 1991.
- [59] C. M. Rhie and W. L. Chow, "Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation," in *AIAA Journal*, vol.21 (11), pp.1525-1532, Nov 1983.
- [60] S. A. Morsi and A. J. Alexander, "An Investigation of Particle Trajectories in Two-Phase Flow Systems," in *Journal of Fluid Mechanics*, vol.55 (2), pp.193-208, Sept 1972
- [61] A. Haider and O. Levenspiel, "Drag Coefficient and Terminal Velocity of Spherical and Nonspherical Particles," in *Powder Technology*, vol.58, pp.63-70, 1989
- [62] V. C. Liu, S. C. Pang, and H. Jew, "Shear grad in flows of almost free molecules," in *Physics of Fluid*, vol.8, pp.788, 1965

- [63] H. Ounis, G. Ahmadi, and J. B. McLaughlin, "Brownian Diffusion of Submicrometer Particles in the Viscous Sublayer," in *Journal of Colloid and Interface Science*, vol.143 (1), pp.266-277, 1991.
- [64] P. G. Saffman. "The lift on a small sphere in a slow shear flow," in *Journal of Fluid Mechanics*, vol.22, pp.385-400, 1965.
- [65] L. Talbot et al. "Thermophoresis of Particles in a Heated Boundary Layer," in *Journal of Fluid Mech*, vol.101 (4), pp.737-758, 1980.
- [66] A. Li and G. Ahmadi, "Dispersion and Deposition of Spherical Particles from Point sources in a Turbulent Channel Flow," in *Aerosol Science and Technology*, vol.6, pp.209-226, 1992.
- [67] Friedlander, S. K., (1977). *Smoke, dust and haze: fundamentals of aerosol behavior*. John Wiley & Sons, Inc.
- [68] Hesketh, H. E., "Fine particles in gaseous media," *Lewis Publishers, Inc*, 1986.
- [69] E. Whitby, F. Stratmann, and M. Wilck, "Fine Particle Model (FPM) for *Fluent* manual," in *FPM 1.0.4*, 2003.
- [70] E. Whitby, "Modal Aerosol Dynamics Modeling," *PhD. Thesis*, Dept. of Mech. Eng., Univ.of Minnesota, 1991.
- [71] P. J. Roache, "Verification and Validation in Computational Science and Engineering" in *Albuquerque*, Hermosa Publishers, New Mexico, 1998.
- [72] P. J. Roache, "Fundamentals of Computational Fluid Dynamics", in *Albuquerque*, Hermosa Publishers, New Mexico, 1998.
- [73] M. J. Ali and H. V. Georgios, "Reynolds Stress Model in the Prediction of confined turbulent swirling flows", in *Journals of Fluid Engineering*, Vol. 128, issue 6, pg. 1377-1383.
- [74] S. Kirpekar and D. B. Bogy, "A study on the efficacy of Flow Mitigation Devices in Hard Disk Drives," in *IEEE Transactions on Magnetics*, vol.42, no.6, June 2006.