Modeling of Hurricanes and Vortices using Direct Simulation Monte Carlo Method

Julian Ngan,
Fernando Alonso-Marroquín,
Sergio-Andres Galindo-Torres
MoSCoS, School of Physical and Mathematical Sciences,
The University of Queensland, St. Lucia, Qld. Australia 4067
s4053667@student.uq.edu.au

February 10, 2009

Abstract

This report investigated the possibility of simulating various aspects of hurricanes and vortices using simplified approach, namely Newtonian particle motion and direct simulation Monte Carlo (DSMC) method. It was found that the single particle Newtonian approach can reveal insights and aspects of real hurricane, such as how the Coriolis force interacts with air at different latitudes. In the Newtonian approach, primary and secondary forces were identified, the particle's trajectory were then computed by integrating the equations of motion.

The DSMC approach uses similar mechanics to compute particle's motions, however it is a more physically realistic attempt to model the physical behavior of hurricane and other types of vortices as simulation uses a large population (order of 10^6) of particles. The inertial forces of particles were also modeled in this approach. Interaction of the particles were modeled using the standard DSMC regime, where the outcome of collisions were predicted based on Kinetic Theory of Gas and the Maxwell's distribution. Although the simulated result thus far do not resemble expected physical vortex phenomena, the author strongly believes that the DSMC approach can ultimately model vortex phenomena accurately. However this requires further investigation on cell grid structures, boundary conditions, implementation in 3D and access to more powerful computer cluster.

1 Introduction

An advanced computational model of hurricanes has the capabilities to predict detailed behavior, structure and trajectory of the hurricanes through detailed input of initial and boundary conditions. Such model would likely be an integrated part of an advanced numerical weather forecasting system. These systems often requires real-time high resolution data of the atmosphere and ocean over a very large area of geographical locations. The Australian Bureau of Meteorology employs such a system, namely the Operational Numerical Analysis & Prediction System. This system consist of many models that focus on various aspects of weather behavior at a range of time-scales. Development of such system and models are beyond the scopes of this report. [2] contains detailed description of Numerical Weather Prediction Products used by the Australian Bureau of Meteorology.

The focus of this report will be towards using relatively simple numerical routines (the DSMC method), a commercial Computational Fluid Dynamics (CFD) package (CFD-ACE) and analytical equations to obtain description of the flow field of cross section of a hurricane and simple vortex found in nature. The DSMC and analytical approach were motivated as the implementation of continuum Navier-Stokes equations to model vortex and weather phenomena were cumbersome to implement and computationally expensive. It is hoped that some aspects of physical phenomena of vortices can be modeled using these approaches. The commercial CFD package used are call CFD-ACE (Fastran), it's primary job is to varify results generated with the DSMC method. The CFD-ACE package solves the continuum Navier-Stokes equations, it provide user friendly graphical user interface to generate simulation grids, implantation of initial & boundary conditions, solves the problem in steady state or transient and provide powerful solution viewing tools.

1.1 The Hurricane

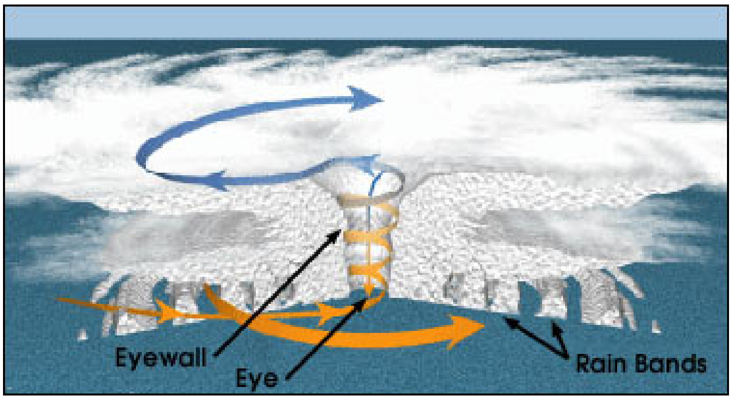


Figure 1.1: Structure of a hurricane [5].

Hurricanes also known as Cyclones & Typhoon, are system of intensive cyclonic storms that forms over warm tropical oceans (see Figure 1.1). It is the most energetic weather behavior seen on Earth. Hurricanes has a typical diameter of about 600km with a central sea level pressure of about 95kPa. The sustained wind speed of hurricanes are at least 104km/hr, if the speed is below this value then the cyclonic storm systems are known as tropical storms. Due to the naturally catastrophic force and large scale of hurricanes, the exact physical formation process are not well understood and still currently debatable. However, there are well defined conditions for which formation of hurricanes becomes possible.

Conditions required for hurricane formation [7]:

First Extensive ocean area with surface temperature greater than 27°C with depth more than 50 meters. This is because ocean at this temperature has significant evaporation to sustain unstable convection and thunderstorms. Hence hurricanes only occurs near the tropics where water temperature are high enough.

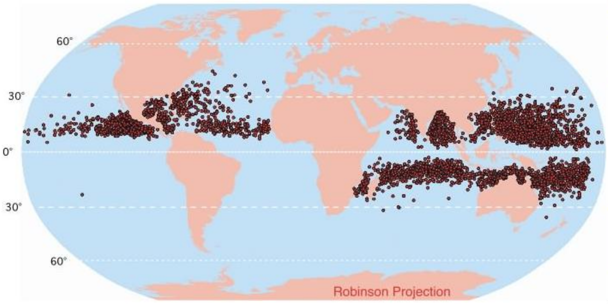
Second The Coriolis force needs to be significant enough to deflect winds initially directed towards the low pressure center to initiate circulation. Hence almost all hurricanes forms at an latitude of at least 5° away from the equator.

Third The vertical wind shear (velocity gradient) must be small (less than 40km/hr), otherwise circulation will be disrupted and hurricane structure cannot form.

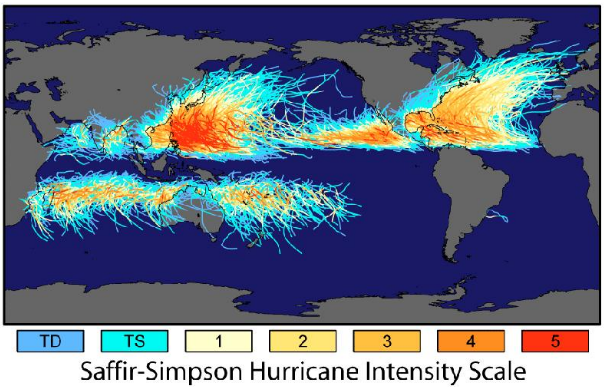
Forth A rapidly cooling atmosphere with respect to height, such that the instabilities encourages thunderstorm activity. It is the thunderstorm activity which allows the heat stored in the ocean waters to be liberated for the tropical cyclone development [2].

Fifth Relatively moist layers near the mid-troposphere (5 km altitude). As wet mid levels are needed to allow conduction for continuous development of widespread thunderstorm activity [2].

While the above conditions are necessary for hurricane formation, it does not mean hurricane or cyclonic disturbance will necessarily form even when these conditions were met. Figure 1.2 shows the formation locations and trajectories of past hurricanes existed on Earth. Notice in Figure 1.2a that hurricanes do not form over the southeast Pacific Ocean, South Atlantic Ocean, and coast of northern Africa because in these regions the cool ocean temperatures restrict formation[1].



(a) Formation locations



(b) Trajectories

Figure 1.2: The formation locations and trajectories of hurricanes from 1950 to 2000 [6].

The flow field of hurricanes are complex, when hurricanes become fully developed, an eye and series of rain bands become visible. Figure 1.3 shows a vertical cross section of a hurricane, demonstrating the structure and direction of flow field within the section. It can be seen in Figure 1.3 that besides the spiral motion seen from the top in Figure 1.1, the eye has a stable downdraft giving the eye a clear sky. In the rain cloud bands are strong updrafts and in between these bands are high turbulent flow with slight down drafts [2].

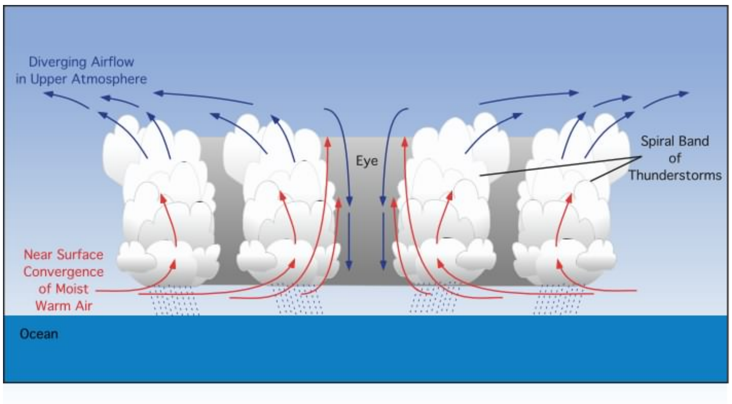


Figure 1.3: The flow field of a hurricane [6].

[1]There are also other hypothesis which suggest no formation in these regions [6]

2 Newtonian single particle hurricane

Some aspects of a hurricane can be predicted or revealed by considering how a single particle of arbitrary size behaves when subjected into the conditions of a hurricane. In a hurricane, the primary driving forces for this arbitrary particle are *buoyancy* and *radial pressure gradient*. The secondary forces are inertial forces due to Earth's rotation, namely the *Coriolis* and *Centrifugal* forces. These secondary forces are about 3 order of magnitude smaller than the primary forces. Although the secondary forces are very small, it plays a significant role in the trajectories over a long distance. This is especially true when particles velocities are high, where the Coriolis effect becomes more significant.[2]

2.1 Governing Equations for single particle

Pressure gradient

Taking from the curve fit of measured hurricane pressure data given by [3], the pressure as a function to hurricane center is given by:

$$P(d) = P_0 + (P_e - P_0)\exp(-\frac{a}{d^b}) \tag{2.1}$$

where $\boldsymbol{P_0}$ is the lowest pressure at hurricane center, $\boldsymbol{P_e}$ is the pressure in undisturbed environment and $\boldsymbol{d}$ is the distance to hurricane center. For hurricane Fabian the constants $\boldsymbol{a} = 90.4$ and $\boldsymbol{b} = 1.18$. For hurricane Isabel the constants $\boldsymbol{a} = 30.7$ and $\boldsymbol{b} = 0.96$. This curve fit is also shown in Figure 2.1.

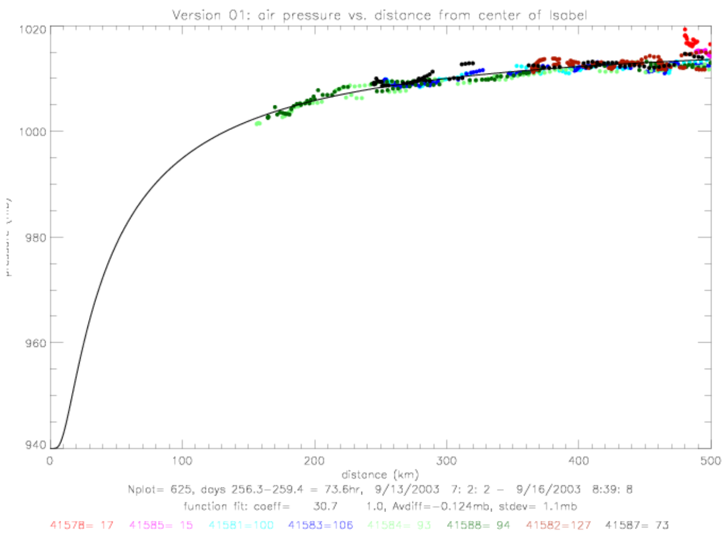


Figure 2.1: Hurricane pressure curve fit in Equation 2.1 [3].

The pressure gradient provide central force for the particles, this force is a function of the pressure gradient given by:

$$\vec{F}_{Pressure} = \frac{dP}{dr} \times dr \times \boldsymbol{A} \tag{2.2}$$

where dP is the pressure differential across the particles, dr is the thickness of the particles, and $\boldsymbol{A}$ is the frontal area of the particle.

[2]Note that in general the Coriolis force is only important for latitude greater than 10 degree North/South.

State properties of air

The state of the atmosphere (temperature, density and pressure etc) versus altitude were determined using the U.S standard atmosphere model 1976, detailed description of the model is available in [4].

Inertial forces

The Coriolis force of a particle traveling in a Earth stationary reference frame is given by:

$$\vec{F}_{Coriolis} = -2m \times \vec{\boldsymbol{\omega}} \times \vec{\boldsymbol{V}} \tag{2.3}$$

where $\vec{\boldsymbol{\omega}}$ is the Earth's rotational vector, m is the particle mass, and $\vec{\boldsymbol{V}}$ is the velocity vector of the particle relative to Earth's stationary reference frame.

The Earth's rotational vector transform into the hurricane center (Earth stationary) coordinate are given by:

$$\vec{\omega} = \left|\vec{\Omega}\right| \times \begin{bmatrix} 0 & \cos(lat) & \sin(lat) \end{bmatrix}$$

where $\left|\vec{\Omega}\right|$ is the magnitude of the Earth rotational vector and lat is the latitude (in degree) of particles.

Buoyancy force

The buoyancy force of the particle is given by [9]:

$$\vec{F}_{Buoyancy} = (\rho_{air} - \rho_{particle}) \times \mathbf{Vol} \times g \tag{2.4}$$

where ρ_{air} is the density of air, $\rho_{particle}$ is the density of the particle, **Vol** is the volume of the particle and g is acceleration due to gravity.

2.2 Integration scheme

A predictor-corrector time integration scheme were used to predicted the particle trajectory. The predictor is Adams-Bashforth Five-Step Method [8]:

$$\vec{\boldsymbol{y}}(i+1) = \vec{\boldsymbol{y}}(i) + \frac{\Delta t}{24}\left[55\vec{\boldsymbol{f}}(i) - 59\vec{\boldsymbol{f}}(i-1) + 37\vec{\boldsymbol{f}}(i-2) - 9\vec{\boldsymbol{f}}(i-9)\right] \tag{2.5}$$

where

$$\vec{\boldsymbol{y}}(i) = \begin{bmatrix} x_i \\ y_i \\ z_i \\ V_{xi} \\ V_{yi} \\ V_{zi} \end{bmatrix} \qquad \vec{\boldsymbol{f}}(i) = \dot{\vec{\boldsymbol{y}}}(i) = \begin{bmatrix} V_{xi} \\ V_{yi} \\ V_{zi} \\ A_{xi} \\ A_{yi} \\ A_{zi} \end{bmatrix}$$

The corrector is Adams-Moulton Four-Step Method [8]:

$$\vec{\boldsymbol{y}}(i+1) = \vec{\boldsymbol{y}}(i) + \frac{\Delta t}{720}\left[251\vec{\boldsymbol{f}}(i+1) - 646\vec{\boldsymbol{f}}(i) + 264\vec{\boldsymbol{f}}(i-1) - 106\vec{\boldsymbol{f}}(i-2) - 19\vec{\boldsymbol{f}}(i-3)\right] \tag{2.6}$$

The accelerations A_{ji} are given by the sum of forces from Equation 2.1, 2.2, 2.3 and 2.4 divided by the mass of the particle:

$$\vec{\boldsymbol{A}}_i = \frac{\vec{F}_{Pressure} + \vec{F}_{Coriolis} + \vec{F}_{Buoyancy}}{m} \tag{2.7}$$

where m is the mass of the particle.

The flow chart of the single particle trajectory simulation is shown in Figure 2.2.

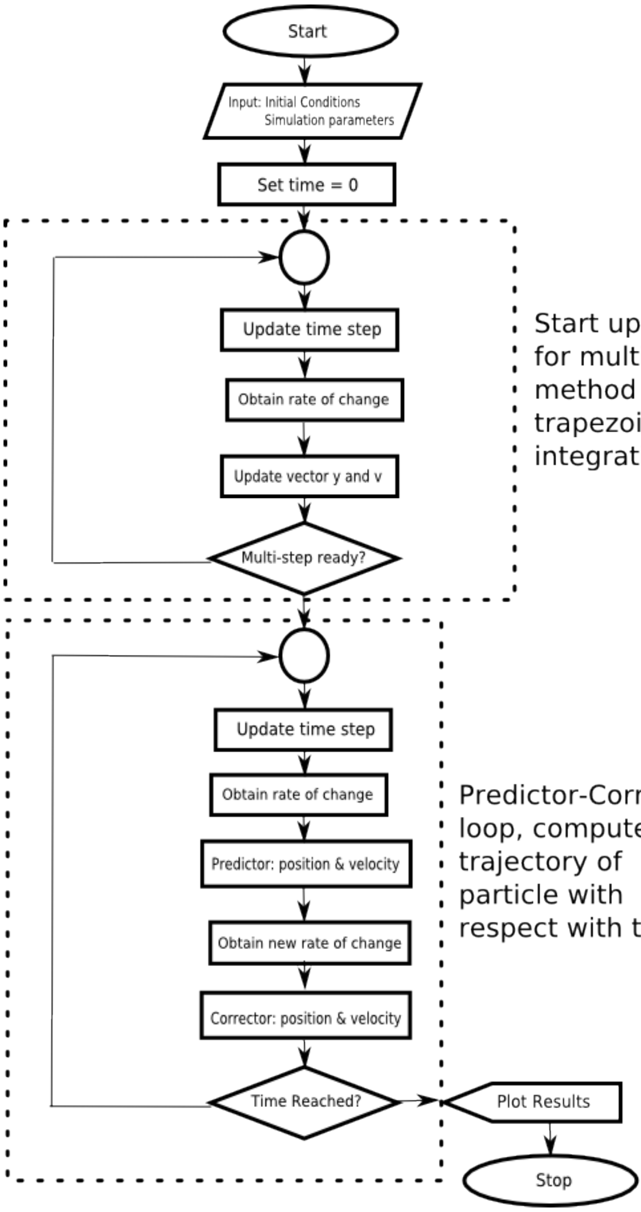


Figure 2.2: Flow chart of hurricane single particle simulation.

2.3 Single particle results

The trajectory of the particle were computed for three latitudes (0°,45° and 90°) with the same atmospheric conditions as expected from a hurricane (see Section 2.1. That is, the temperature, pressure and density of air were kept constant (see Table 1) despite the change in latitude. This were done in order to investigate the Coriolis effect on the particle's trajectory. The initial conditions of the particle were also kept the same for all cases. The simulation parameter for the three cases are summarized in Table 2.

Table 1: The initial atmospheric condition for the single particle simulation at altitude of 100m using US standard atmospheric model

Altitude	Pressure	Temperature	Density
100 m	100.1 kPa	297.5 K	1.21 kg/m^3

Table 2: The simulation parameters for the single particle simulation

Simulation Number	Latitude	Altitude	Initial vel (vx, vy)	Initial pos (x, y)	Simulated time
1	0° N	100 m	$[1, 0]$ m/s	$[2000, 0]$ m/s	300 min
2	45° N	100 m	$[1, 0]$ m/s	$[2000, 0]$ m/s	300 min
3	90° N	100 m	$[1, 0]$ m/s	$[2000, 0]$ m/s	300 min

Latitude $\phi = 0°$

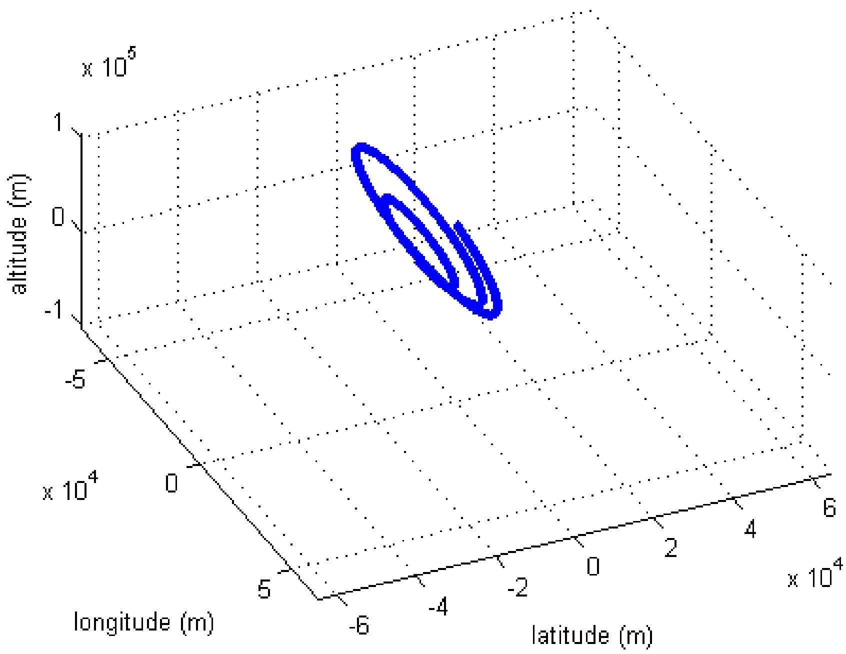


Figure 2.3: The trajectory of particle at latitude=0°

It can be seen in Figure 2.3 that the particle appears to follow a spiral trajectory within a vertical plane (i.e. no y-component of velocity is observed). The particle trajectory is confined within the plane because the Earth's rotational vector at latitude of zero coincide with the y-axis in the Earth stationary reference frame, since the Coriolis force is the cross product between the particle's velocity and the Earth rotational vector, by the right hand rule the Coriolis deflection force should be pointed in the z-direction of changing altitude. Note that the Coriolis effect are relatively smaller when comparing with trajectory at higher latitude (see Results for latitude 45° and 90°). By inspecting Figure 2.3, the particle was initially traveling away from the center, however the hurricane pressure gradient (directed towards center) reverse the particle direction of travel, which the the Coriolis force provide vertical displacements. One can understood why hurricane cannot form at such latitude, because there is no horizontal component of motion that causes circulations. The vertical displacements caused by the Coriolis force will only disrupt a low pressure center.

Latitude $\phi = 45°$

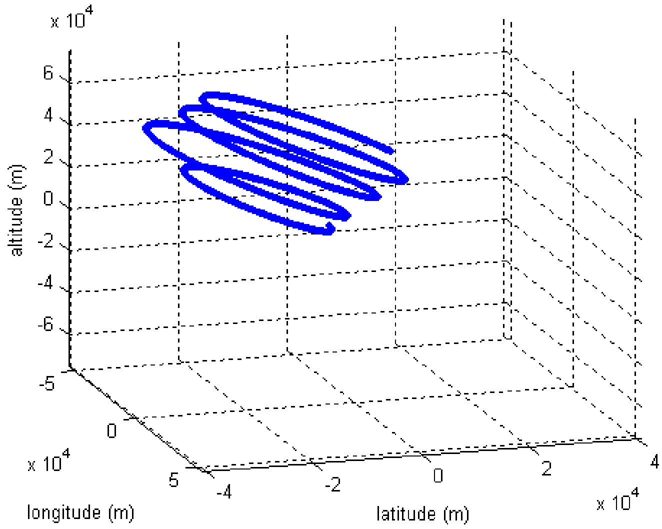


Figure 2.4: The trajectory of particle at latitude=45°

As can be seen in Figure 2.4 that the particle's trajectory is much more similar to one expected for a hurricane. Note that the particle were deflected by the Coriolis force in both vertical and horizontally. The vertical motion causes an updraft, however this updraft should be insignificant compare to updraft caused by temperature and pressure variations (buoyancy force). The vortex motion observed in Figure 2.4 is created when the Coriolis force deflect the particle of motion at right angle to direction of travel[3], while the pressure gradient "attracts" the particle towards the center.

Latitude $\phi = 90°$

At the latitude of 90 degrees[4], the Coriolis effect has negligible vertical component (see Figure 2.5). The Coriolis contribution to vertical component exist because as the particle moves, it latitude changes very slightly. This vertical component of force is negligible relative to buoyancy forces. The particle trajectory follows a "star" like shape. This is primarily because the Coriolis force now acts within the same plane as the pressure gradient. In fact, the Coriolis force is continuously countering the force from the pressure gradient. As the particle is "attracted" towards the center, the Coriolis force counter the attraction in order to conserve the angular momentum[5] of the particle. When the Coriolis force deflected the particle to the extend where the particle velocity is exactly opposite to the pressure gradient (see Figure 2.5), the pressure gradient acts directly to reverse the particle's velocity, hence the oscillation pattern of circulation is seen.

[3]However it the deflection is also not within the same plane of the velocity of the particle, this is because the particle velocity plane is not at right angle to the Earth's rotational vector.
[4]i.e. At the North/South pole
[5]This is the angular momentum with respect to Earth's rotational axis

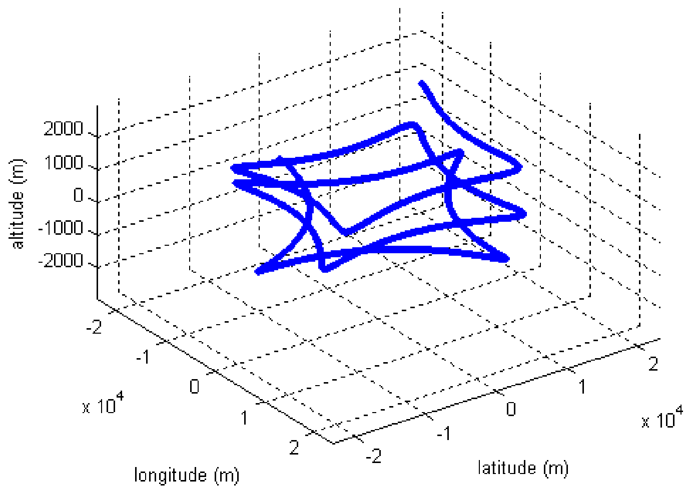


Figure 2.5: The trajectory of particle at latitude=90°

3 Vortex modeling using DSMC method

A definition of the DSMC method given by [1]:

> The Direct Simulation Monte Carlo (DSMC) method is widely used for the modeling of gas flows through the computation of the motion and collisions of representative molecules. Computation at the molecular level is necessary for studies in rarefied gas dynamics (or RGD) because the transport terms in the Navier-Stokes equations are not valid in this flow regime. The essential characteristic of a *rarefied flow* is that the molecular mean free path is not negligible and many applications involve normal and high density flows with very small physical dimensions.

DSMC method potentially have a wide ranges of modeling applications, such as satellite/re-entry vehicle drag estimation (see Figure 3.1), Scram jet engines, mixing & combustion of fuels, and small vortices etc. The remainder of this section describes detailed computational procedures used to model gas vortices.

3.1 2D Vortex DSMC

The main code is responsible for setting up the simulation grids, creating particles with mass, size and velocities in accordance with initial conditions specified. In addition, the main code also performs time integration of particles, handles results from collision loops, storing and plotting simulation results.

The code is written using MATLAB in 2D[6], hence particles are confined in a 2D space deflecting due to Coriolis force and binary collisions as it travels. The initial velocities of the particles are random, however the average (macroscopic) velocity of the particles is zero. Hence if the state properties (pressure and temperature etc) of the particles are uniform across the simulation space, the particles will transport solely due to diffusion. If a property gradient exist (e.g. pressure gradient), then advection will occur. A picture of the simulation space with particles is shown in Figure 3.2.

[6]Note: 3D code was too computational expensive and increases the complexity significantly for the preliminary development of this procedure.

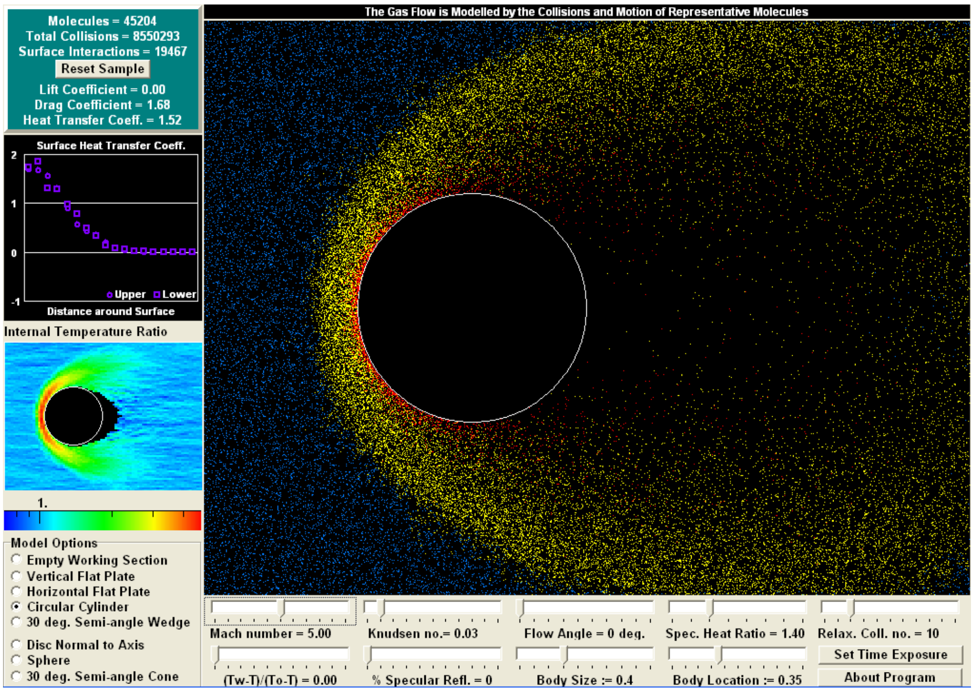


Figure 3.1: A virtual wind tunnel program using DSMC method, computing details of the flow field (e.g. pressure, temperature or velocities) around an object of arbitrary shape [1].

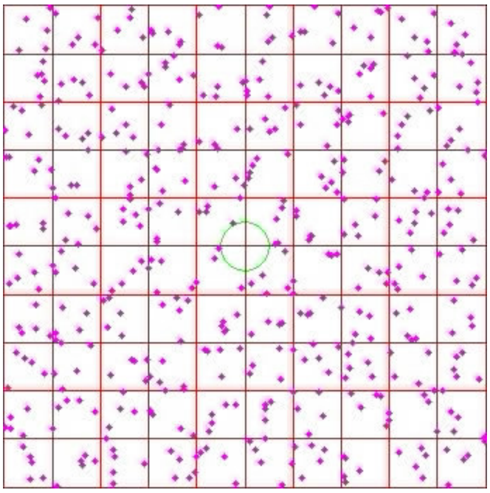


Figure 3.2: Simulation grid, the particles are assigned with random positions and velocities, however means are zero.

Notice in 3.2 that the simulation space is meshed with cells, only the particles within the same cell may collide with one another. Moreover, the particles may allow to collide even if they don't physically touches one another, the only requirement is that the particles must be in the same cell. This is the essence of the DSMC method, it is analogous to finite difference approximation, when the cell size approach the size of the particle, collision can only occur if the particles touches one another. However since the size of the air molecules are extremely small, cell size approaching molecule size is not viable. DSMC approximate the collision process at a molecular level by allowing many particles (order of hundreds particles) to be contained within one cell, then particles within the cell are selected randomly[7] to allow for binary collision. The fraction of particles within a cell to undergo collision is based on the kinetic theory of gas, which depend on the mean free paths, velocities and cell size etc. The outcome of all collisions are random,

[7]Hence the name Monte Carlo

however momentum and energy of are conserved.

The circular inner boundary in Figure 3.2 is an exit boundary, the particle will leave the simulation whenever it reach within the circle. Therefore as particles near the center diffuses into the circle, after some time a density and pressure gradient will develop radially, this creates advection and the motion of particles velocities will be directed towards the center.

Three types of outer boundary has been adapted for the simulation space seen in Figure 3.2:

1. Wall boundary - When particles travels beyond the limits of the simulation space their velocities will be reverse, and their position will be the distance traveled away from the boundary inverted back into the simulation space.

2. Boundary cells regenerated every time step - All the particles in the boundary cell are deleted and regenerated with a state specified, this maintains the boundary condition for the simulation.

3. Boundary cells with state properties maintained constant - Whenever the state property of the cell changes, the routine adjust the boundary cells so the state properties are fixed.

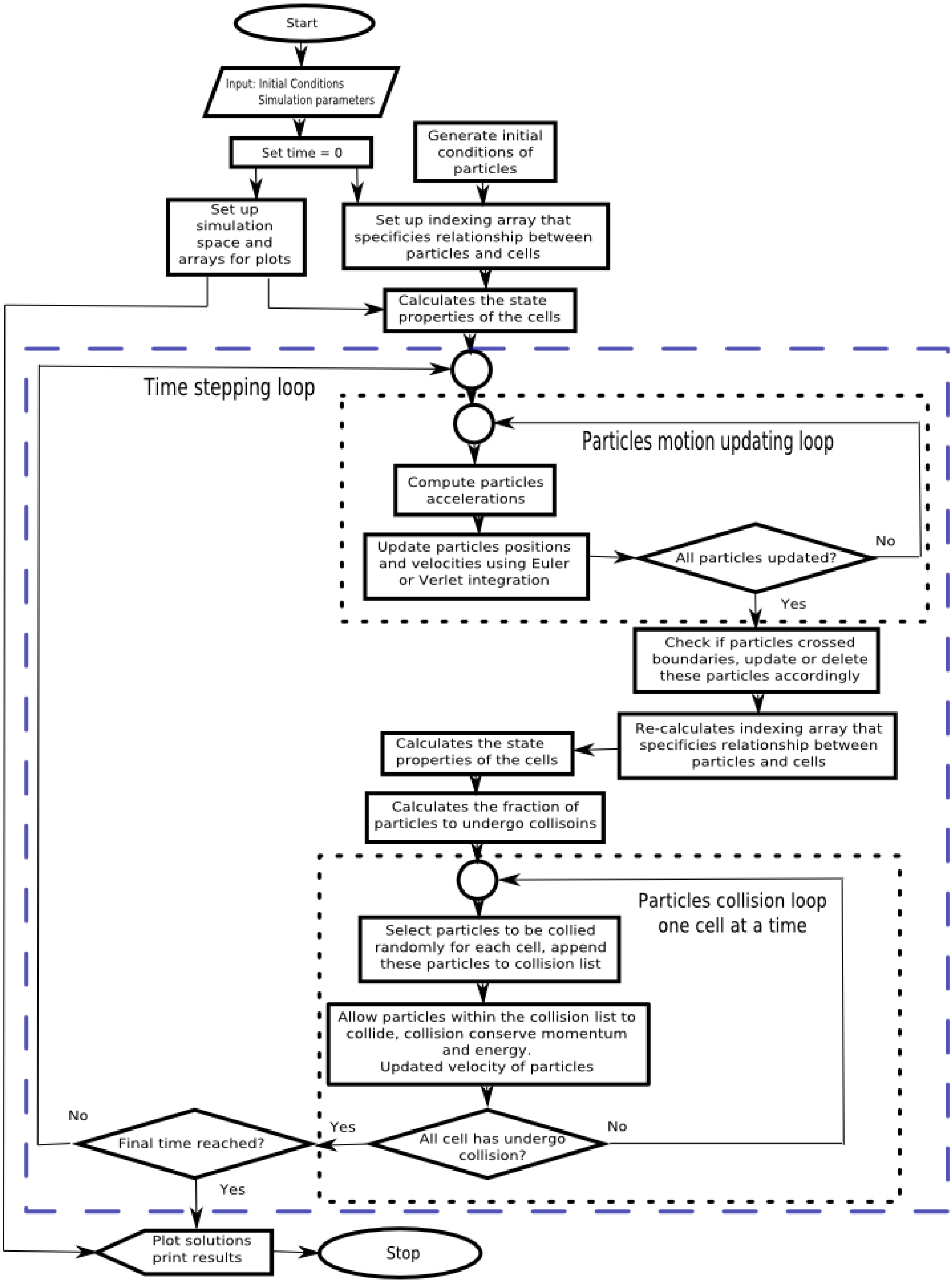


Figure 3.3: The overall flow chart of the 2D vortex DSMC.

3.2 DSMC results

Numerous simulations with different simulation parameters, boundaries, initial conditions and routines were computed. Unfortunately non has computed vortex type flow. However, some of these simulation has converged into a steady state flow. Simulation that converges to steady state will be documented in this report, results for other cases can be provided as request. In addition to these converged cases, a special case have been included. In this special case, particles are attracted towards the center with a central force (the nature of the force is equivalent to that of a hurricane pressure gradient). Normally the particles will only be pushed towards the center due to collision of particles at the outer boundaries when a density (hence pressure) gradient develops as particles disappears when arrived at the circle.

Case 1

The specifications of the simulation Case 1 is summarised in Table 3.

Table 3: The simulation results for Case 1

Simulation case	1
Number of cells	400 cells
Number of particles	10,000 particles
Simulated time	900 seconds
Time step size	0.03
Magnitude of rotational vector	0.05 rad/sec
Boundary Type	Type[8] 3, fixed pressure
Time integration scheme	Euler

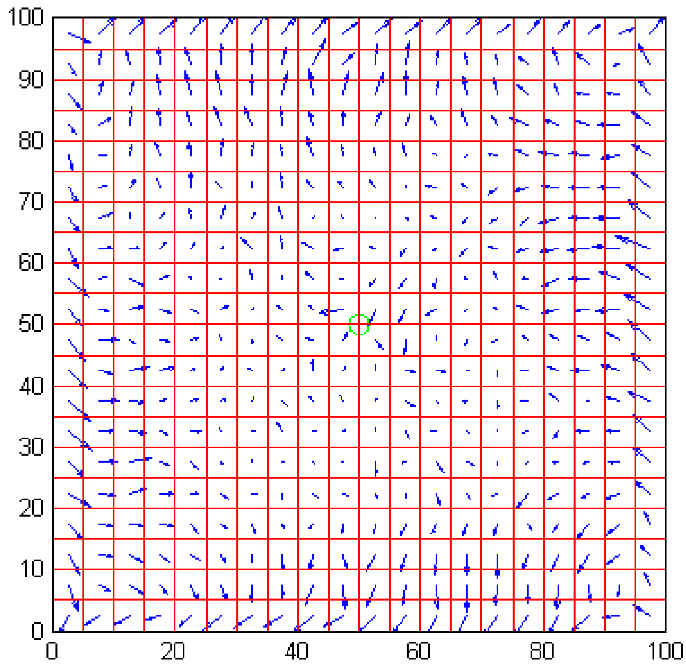


Figure 3.4: The velocity vector field of the cells. The cell velocity is the average velocity of particles within the cell.

The steady state velocity field of simulation Case 1 is shown in Figure 3.4. Some vorticity can be seen about the corners of the simulation space, however no vorticity is seen about the center. The velocities near the outer boundaries are larger than ones near the center, this is opposite of what expected in a vortex rotating about the center. Figure 3.5 shows the density contour of the simulation space, the contour is as expected for vortex, where a density gradient exist and the center cells has lower densities. Note the contour are coarse, improvement can be made by decreasing cell size, and using more particles.

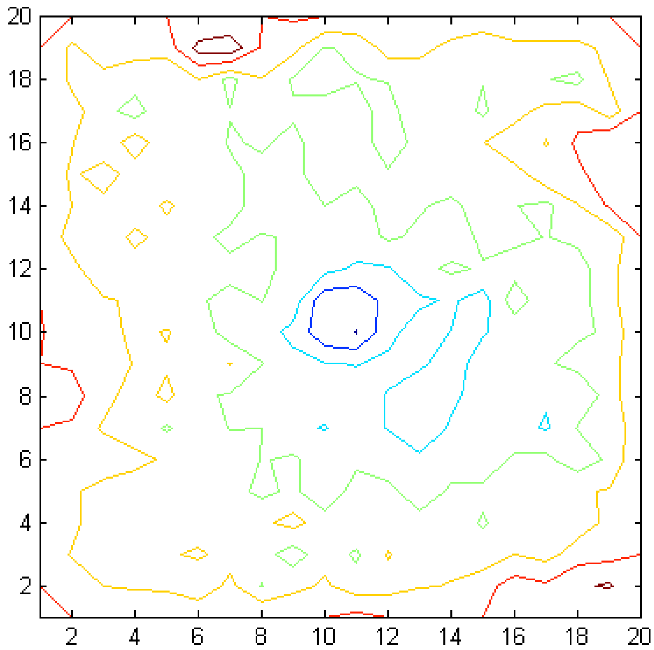


Figure 3.5: The density contour for simulation case 1.

Case 2

The specifications of the simulation Case 2 is summarized in Table 4.

Table 4: The simulation results for Case 2

Simulation case	2
Number of cells	400 cells
Number of particles	10,000 particles
Simulated time	900 seconds
Time step size	0.02
Magnitude of rotational vector	0.01 rad/sec
Boundary Type	Type[9] 3, fixed pressure
Time integration scheme	Verlet

The velocity vector field for simulation Case 2 is shown in Figure 3.6. Notice this time the velocities are high near the center and low at outer cells (expect outer most boundary cells). This is almost an improvement, but again there is no vorticity about the center. Note that four vorticity appears about the four corners.

Figure 3.7 shows the density contour of simulation Case 2. Notice this time the contour is much

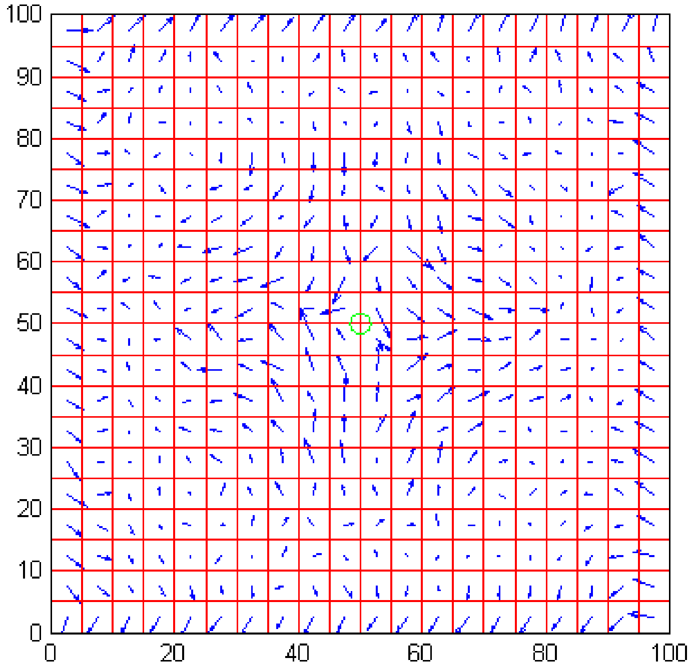


Figure 3.6: The velocity vector field of the cells for simulation case 2. The cell velocity is the average velocity of particles within the cell.

more smoother than Case 1 Figure 3.5. This indicates possibly a better simulation regime, the flow field is much more uniform and steady. It seems confident that if the cells were circular and more refined grid will result in an almost perfect hurricane/vortex matching contour.

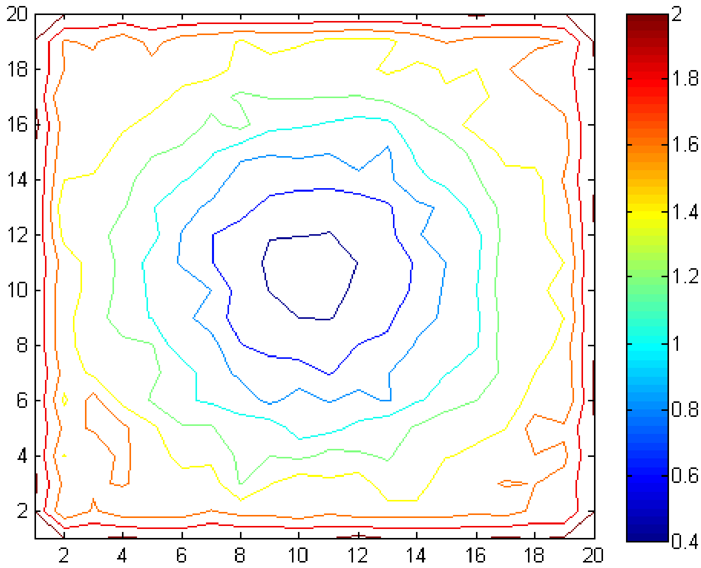


Figure 3.7: The density contour for simulation case 2.

Case 3

Note that this is the special case where the central force applies as discussed in the beginning of this section. The specifications of the simulation Case 3 is summarized in Table 5.

Table 5: The simulation results for Case 3

Simulation case	3
Number of cells	100 cells
Number of particles	500 particles
Simulated time	200 seconds
Time step size	0.05
Magnitude of rotational vector	0.05 rad/sec
Boundary Type	Type[10] 1, wall
Time integration scheme	Euler

The initial position of the particles are shown in 3.8, these are randomly generated with uniform state properties across simulation space. After about 3-4 seconds, a strong vorticity develops about the center (see Figure 3.9). However as the simulation propagates, these particles leaves the center and vorticity vanishes. This is due to the fact that the particles are following a star shape like trajectory similar to that shown in Figure 2.5, and when the particles arrive at the corners of the star shape, they reach the wall boundary that deflects them to extend where direction of travel may reverse. Since the numbers of particles is not great enough to damp out such "star shape" oscillation, the vorticity is destroyed. Moreover, if there were more particles on the outer cell in Figure 3.9, then these rotating particles will be force to stay near the center and vorticity may continues to develops.

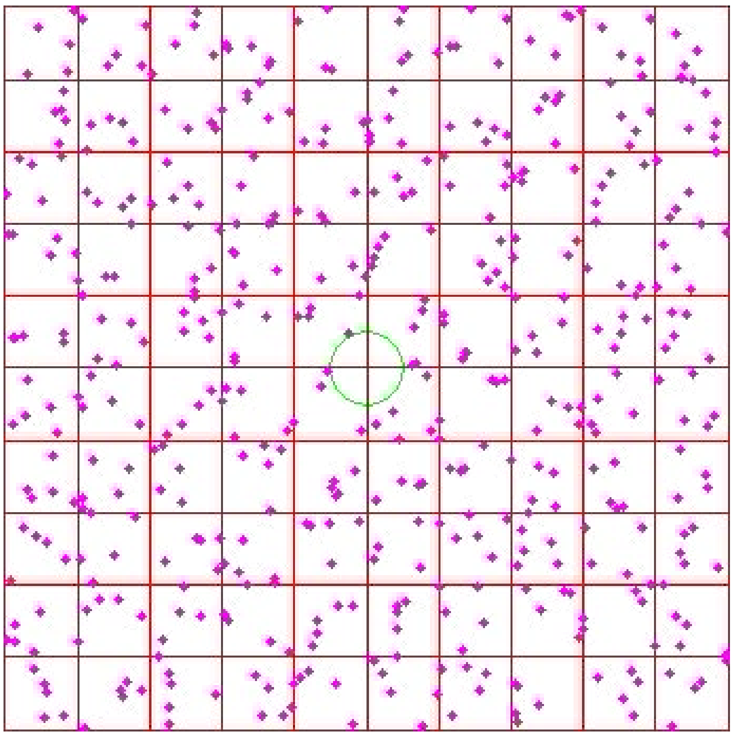


Figure 3.8: Initial position of particles for simulation Case 3.

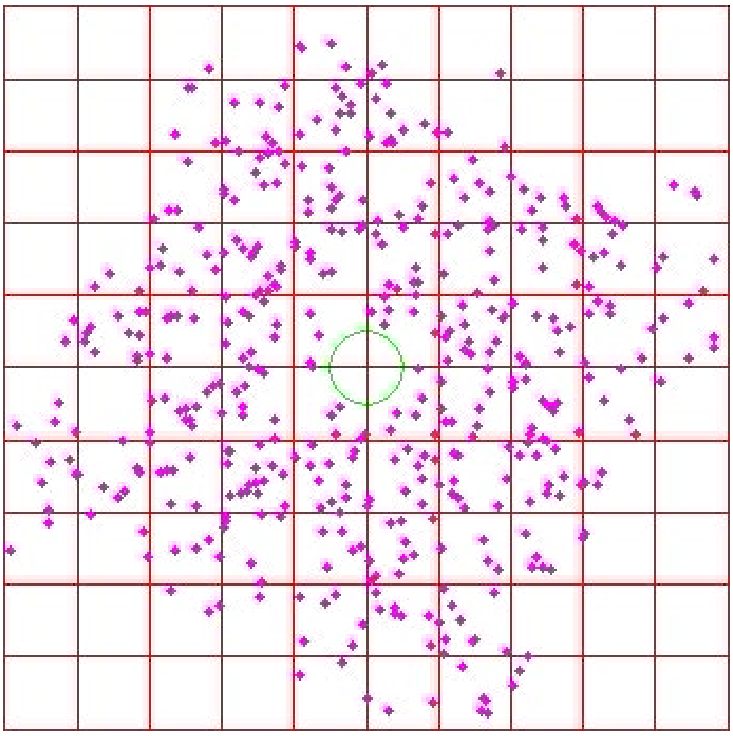


Figure 3.9: Position of particles for simulation Case 3, after about 4 seconds rotation develops.

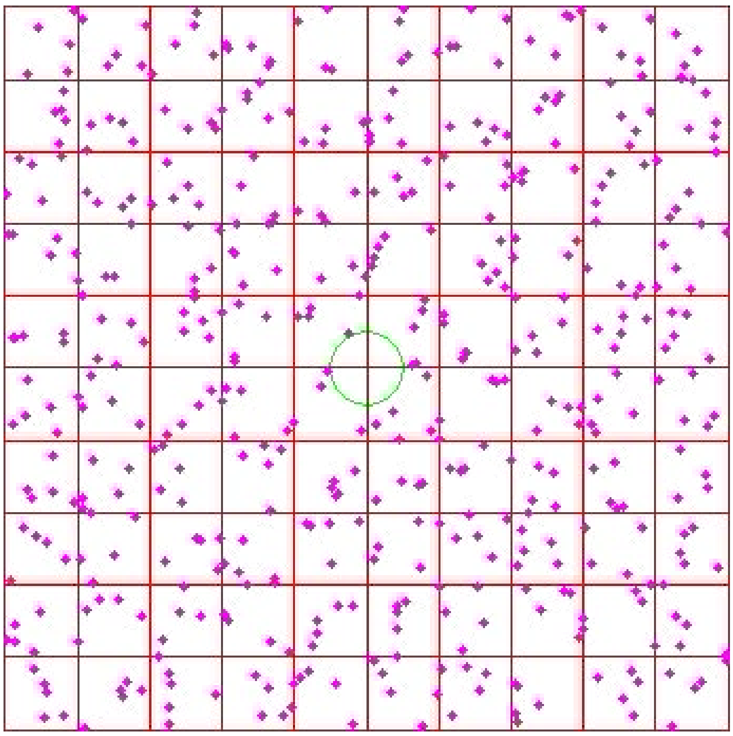


Figure 3.10: Position of particles for simulation Case 3 at the end of simulation, the vorticity seen in Figure 3.9 totally vanishes. Notice there is less particles in the space as comapared with Figure 3.8 as some particles entered the center hole.

4 Conclusion

This report has studied the structure of a hurricane briefly, then largely simplified simulations were used to investigate some physical phenomena observable in a hurricane and vortices. The main aspects the single particle model revealed was how the Coriolis affects the trajectory of the particle with respect to the latitude. Vorticity can of the particle is greatest at mid-latitude. While vorticity exists at the poles, the Coriolis force counteract with the pressure gradient, hence rotation were not smooth and vorticity were lowered.

The implementation of the DSMC method were done on a 2D rotational non-inertial reference frame. The particles were subjected to Coriolis force and collisions. The boundaries were developed such that the outer boundaries has constant properties (similar to a vortex or hurricane where at the outer most boundaries the state properties are relatively constants. The eye of hurricane or the drain (similar to a bathtub drain) were modeled such that it is expected vortex will appear. However, after many attempts and fine tuning of the simulation codes, no vortex have been observed. There are few speculation as to why vortex were not observable:

1. Vortex are 3-dimensional physical behavior, hence a 2-dimensional model cannot demonstrate vortex.
2. The number of simulation particles used were insufficient to model gas behavior realistically to produced vortex type flow.
3. The grid cell size were too large, so vorticity were destroyed due to unrealistic physical relationship between cells.
4. Vortex do not exist in rarefied type flow, hence only when the continuum assumption is valid that vortex will be observable. This mean there may be a correlation between Knudsen number and vorticity.
5. The shape of the simulation grid and boundaries do not encourage vortices.

The author believes there should be valid reasons to explain why vortex were not observable, and that if these reasons were identified then the simulation code may be modified accordingly to obtain vortex. Few improvement to the simulation are now suggested:

1. Using a circular grid.
2. Implement DSMC method in more efficient language than MATLAB, such as C++.
3. Run the simulation using a supercomputer. Objective primarily to increase the number of particles and decrease cell size.
4. Introduce shear flow or initial vorticity into the initial conditions.
5. Improving the collision scheme by allow multiple particles to undergo a single collision, however the probability of such collisions must follow that to kinetic theory of gas.
6. Introduce other dynamics of particles such as rotations and vibrational modes.
7. Using adaptive time stepping and cell size for faster computations.

Continuum CFD code CFD-ACE has been implanted to create a 2D planer vortex. This was initially done in order to check results using DSMC method. At this stage the CFD-ACE code diverges and give an error message for the same conditions specified in the DSMC method.

Further, investigation and study of the CFD-ACE package is needed to learn why results diverges and how to apply the boundary conditions correctly so that sensible results could be obtained.

Analytical solutions using the stream functions from potential flow theory have produced 2D flow field that describe bathtub vortex and tornado type flow accurately. The analytical approximation is accurate up until near the center of rotation. Applying the stream functions in 3D will be the next aim in order to describe the flow filed more completely. These technique may be used to verify the continuum and DSMC approach when they are more fully developed.

References

[1] G.A Bird. Direct Simulation Monte Carlo Method. http://www.aeromech.usyd.edu.au/dsmc_gab/, July 2008.

[2] Bureau of Meteorology Commonwealth of Australia. Operational numerical analysis & prediction system. http://www.bom.gov.au/nmoc/ab_nmc_op.shtml, 2009.

[3] Jan Morzel. Drifter Data in Hurricane Momentum Equation, during Fabian & Jeanne (2003) and Frances & Isabel (2004). http://www.cora.nwra.com/~morzel/drifters.momentum.equation.html, June 2008.

[4] NASA. *U.S. Standard Atmosphere, 1976.* U.S. Government Printing Office, Washington, D.C., 1976.

[5] NOAA. Hurricane structure. http://earthobservatory.nasa.gov/Library/Hurricanes/Images/hurricane_structure.jpg, August 2005.

[6] Michael Pidwirny. Tropical Weather and Hurricanes. http://www.physicalgeography.net/fundamentals/7u.html, November 2008.

[7] Richard J.C. Roger G.B. *Atmosphere, Weather & Climate.* Methuen, 1987.

[8] Timothy Sauer. *Numerical Analysis.* Pearson, Addison Wesley, 2006.

[9] F.M. White. *Fluid Mechanics.* Mc Graw Hill, 2007.