# Modeling and Simulating Flow Phenomenon using Navier-Stokes Equation

**Hyunjun Min, Jiho Shin, Jaeho Choi, and Hohyun Lee**
Data Science Lab, Paul Math School
12-11, Dowontongmi-gil, Cheongcheon-myeon, Goesan County, Republic of Korea
alsguswns1123@gail.com, hizibu7@gmail.com, jaeho.choi@pmath.org, hohyun.lee@pmath.org

## Abstract

Due to drastic development in computing environment Computational Fluid Dynamics (CFD) area can be simply modeled and simulated for better understanding of complex systems. So it enables many analysis of complex flux. In this research induce Navier-Stokes equation's mathematical, Fluid dynamical content and examine using method in CFD and compared result of CFD and result of coding. lastly induce governed equation at model's conditions.

## Keywords
Computational fluid dynamics, Navier-Stokes Eq., Flow phenomenon, Fluid dynamics

## 1. Introduction

Navier-Stokes Equation is governing equation of Fluid dynamics and used importantly on many areas like Mechanical engineering, Mathematics, Fluid dynamics, civil engineering. In this equation's case can signify perfect flow because it gets variables about all of value which is about fluid flow. So this equation always importantly come to the fore on Fluid dynamics. But Navier-Stokes equation get variable about all of value, so find 3-D Navier-Stokes equation solution problem is registration on Millennium problem by Clay Mathematics Institute (CMI). In beginning Navier-Stokes equation be promising to peoples because of possibility to calculate every fluid flow academically. But in reality applicable models are only few. Now due to development of computer, copious calculation (human impossible to calculate) enable to calculate. So acceptable model's area increase and now pioneer to Medical science area like analysis of vascular flow.

Cholesterol which ingest with nourishment adhere to blood vessel wall and caused many adult diseases like Myocardial infarction and Atherosclerosis. Cholesterol grow vertically due to Cholesterol reduce area.

According to Bermouilli equation, area and velocity are direct proportion. So If cholesterol adhere to blood vessel wall, area become smaller than normal blood vessel's cross-section area and then velocity become faster. Due to fast velocity, Cholesterol's edge gets hurt. So scab created on the cholesterol and area become smaller than before. Cholesterol widely and constantly increase due to the blood retention which caused by the gap created by cholesterols and blood vessel wall at the backside of the cholesterols.

We started this study to analysis about various flow phenomenon with Navier-Stokes equation on the fluid dynamical viewpoint and analysis formula on the mathematical viewpoint in this situation.

In this paper analysis Navier-Stokes equation which simplify basic blood flow to modeling similar to cholesterol interrupter at the blood vessel wall. And we run CFD with correct condition. After that we compare analysis value which analysis by Navier-Stokes equation, CFD and cording (use programing language C). And this is the final purpose and conclusion of this research.

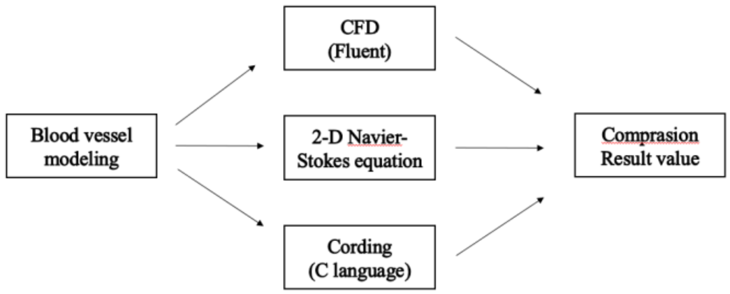


**Fig 1 Study method**

# 2. Body

## 2.1 Precedence study for blood flow analysis

### 2.1.1 Total differential and partial differential

Lower formula is very important in fluid dynamics.

$$\frac{du}{dt} = \frac{\partial u}{\partial t} + u\,\frac{\partial u}{\partial x} + v\,\frac{\partial u}{\partial y} + w\,\frac{\partial u}{\partial z}$$

Like this formula is directed by process of proof relationship between total differential and partial differential mathematically.

Total differential and partial differential is differential of higher than two variable function like $z = f(x, y)$. So it is differential of type of function which has many variables. If analysis relationship mathematically, normal function $z = f(x, y)$ is formula which made up with x, y. So $z = f(x, y)$ can transformation like

$F(x, y) =$ formula about x + formula about y + (formula about x)(formula about y)

So $z = f(x, y)$ change like lower.

$$z = F(x,y) = f(x) + g(y) + h(x)k(y)$$

At this time conduct partial differential to get $\frac{\partial z}{\partial x}$ is like lower formula.

$$\frac{\partial z}{\partial x} = \frac{\partial f(x)}{\partial x} + \frac{\partial g(y)}{\partial x} + \frac{\partial h(x)k(y)}{\partial x}$$

At this point $\frac{\partial z}{\partial x}$ is partial differential about x, so if calculate partial differential, y treated like constant number. So equation like lower came out.

$$\frac{\partial z}{\partial x} = \frac{\partial f(x)}{\partial x} + k(y)\frac{\partial h(x)}{\partial x}$$

And then upper formula become like lower formula

$$\frac{\partial z}{\partial x} = f(x) + k(y)h(x)$$

and sameness, $\frac{\partial z}{\partial y}$ become like lower formula

$$\frac{\partial z}{\partial y} = g(y) + h(x)k(y)$$

Next is arrange method use total differential.

Different with upper formula, this formula differential totally. So if z is $z = f(x, y)$, $\frac{dz}{dt}$ is developed like lower formula.

$$\frac{dz}{dt} = \frac{df(x)}{dt} + \frac{dg(y)}{dt} + k(y)\frac{dh(x)}{dt} + h(x)\frac{dk(h)}{dt}$$

At this time, $\frac{df(x)}{dt}$ can't calculate. So use chain rule to arrangement formula like lower formula.

$$\left[\frac{df(x)}{dx} + \mathrm{k(y)}\frac{dh(x)}{dx}\right]\frac{dx}{dt} + \left[\frac{dg(y)}{dy} + h(x)\frac{dk(y)}{dy}\right]\frac{dy}{dt}$$

So dz become

$$dz = [f(x) + k(y)h(x)]dx + [g(y) + h(x)k(y)]dy$$

and at this time deduction partial differential equation at upper formula like

$$\frac{\partial z}{\partial x} = f(x) + k(y)h(x) \text{ and}$$
$$\frac{\partial z}{\partial y} = g(y) + h(x)k(y).$$

And if substitute this equation inside the total differential parentheses, it become like lower formula and we can know about relationship between total differential and partial differential.

$$\frac{dz}{dt} = \frac{\partial z}{\partial x}\frac{dx}{dt} + \frac{\partial z}{\partial y}\frac{dy}{dt}$$

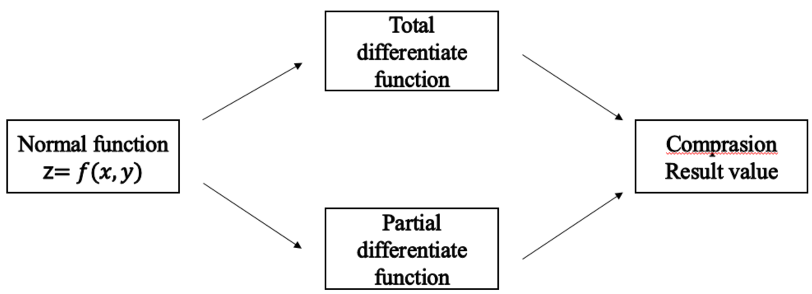


**Fig 2 Process of prof relation between total differential and partial differential**

### 2.1.2 Fluid dynamical approximation

Flows are complicate exercise which creation due to particles exercise to many directions. And conservation equation about exercise have many variables. So partial difference must use in fluid dynamics and due to properties like fluid, in fluid dynamics use partial differential is more simplicity than using total differential unlike other material dynamics.

In fluid dynamics, we can draw out the equation

$$\frac{du}{dt} = \frac{\partial u}{\partial t} + \frac{\partial u}{\partial x}\frac{dx}{dt} + \frac{\partial u}{\partial y}\frac{dy}{dt} + \frac{\partial u}{\partial z}\frac{dz}{dt}$$

By applying the relation of total differential and partial differential due to it uses 'u(x, y, z, t)' instead of general function like z(x, y) which used in mathematics.

In fluid dynamics, because of 'u, v, w' each means the velocity to the direction 'x, y, z' so $\frac{dx}{dt} = \mathrm{u}$, $\frac{dy}{dt} = \mathrm{v}$, $\frac{dz}{dt} = \mathrm{w}$ is done.

Then, the equations the upper becomes the equation like lower

$$\frac{du}{dt} = \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}$$

And use in fluid dynamics importantly.

Most dynamics uses normal differential and total differential because it describes movement at a perspective of a single particle. However, in fluid dynamics, because fluid has a similar meaning as the group of multiple particles, it is hard to describe fluid in a perspective of a single particle. So, fluid dynamics follow the ways of Euler coordinate systems which sets a single section at a perspective of an observer and calculates the physical amount of fluid that comes and goes through the section. At this time, the set exact section is called 'control volume' and we can define flow with considering the inner changes of flow and the input.

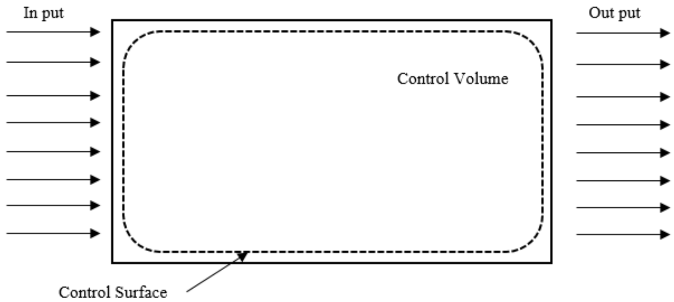


**Fig 3 Shape of control volume**

Equation about intensity amount's change follow time inside control volume is lower formula.

$$\frac{d}{dt}\left(B_{syst}\right) = \frac{d}{dt}\left(\int_{CV} \beta\rho dV\right) + \int_{CS} \beta\rho(\vec{v}\cdot\vec{n})dS$$

Now left side is indicated that system in B of variation value and right side of first term is B of total amount that out flowed and in flowed from control volume. Now the vector in second slot of right side is consider both of speed of control volume and fluid. So we variation for doing continuity equation.

$$0 = \frac{\mathrm{d}}{\mathrm{dt}}\int_{\mathrm{CV}} \rho \mathrm{dV} + \int_{\mathrm{CS}} \rho(\vec{\mathrm{v}}\cdot\vec{\mathrm{n}})\mathrm{dS}$$

And represent differential shape to do gauss divergence theorem.

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

We can get continuity equation that control volume itself and fluid move by the same time, also considering speed of fluid and fluid of control volume speed.

$$\frac{d(m\,\vec{v})}{dt} = \frac{d}{dt}\left(\int_{CV} \vec{v}\rho dV\right) + \int_{CS} \vec{v}_f\rho\,(\overrightarrow{v_{tot}}\cdot\vec{n})dS$$

### 1.3 Surface force of control volume

In the case of surface forces, it is the force due to the stress acting on the surface. At this time, the sum of the shear stress $\tau_{ij}$ related to the hydrostatic pressure and the velocity gradient is the sum of the total surface forces of the surface forces. Unlike the velocity with three components, the shear stress $\tau_{ij}$, the stress $\sigma_{ij}$, and the strain $\epsilon_{ij}$ are all nine tensors, and each component has two subscripts, unlike the one with a subscript. The suffix indicates the direction of the surface to which it belongs, and the suffix j indicates the direction in which the force acts.

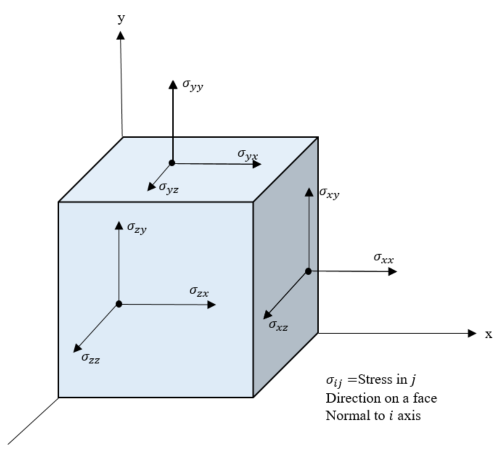


**Fig 4 stress on x, y, z faces**

For example In an instance of momentum, If input value is $\sigma_{zz}dydz$, out value is become control's volumes inner change value like lower formula.

$$\left[\sigma_{zz} + \frac{\partial \sigma_{zz}}{\partial x} dx\right] dydz$$

Therefore in an instance of add all of available surface force about x direction which affected on the three faces. And it represented proportion between surface force and volume of elements.

$$\mathrm{d}F_{x,surf} = \left[\frac{\partial}{\partial x}\sigma_{xx} + \frac{\partial}{\partial y}\sigma_{yx} + \frac{\partial}{\partial z}\sigma_{zx}\right] \mathrm{dxdydz}$$

Upper formula can transform like lower formula for pressure and shear stress.

$$\frac{dF_x}{dV} = -\frac{dp}{dx} + \frac{\partial}{\partial x}(\tau_{xx}) + \frac{\partial}{\partial y}(\tau_{yx}) + \frac{\partial}{\partial z}(\tau_{zx})$$

Since each term in the parentheses in the equation for viscous force represents the divergence of the stress component vector acting on the x, y, z faces, the above equation can be expressed as divergence form.

$$(\frac{dF}{dV})_{viscous} = \nabla \cdot \tau_{ij}$$

## 2. Model's governed equation

### 2.1 Navier-Stokes equation

If Navier-Stokes equation's general value has discovered, we can know beforehand about flow's properties which comes to surround of specific object with constant pressure and velocity. So it has tremendous development prospect in the areas like civil engineering, mechanical engineering, chemistry, aeronautical engineering etc…Also if Navier-Stokes equation become generalization, it is possible to get general value of turbulent flow Which can analysis depend on experience.

In an instance of Navier-Stokes equation is based on Newton's $2_{nd}$ law of motion (F = ma). In addition, there are various forms because it takes a constant value for various terms according to the fluid or a process of making 0 according to the condition.

Navier-Stokes equation's popular form is represented by Einstein's convention.

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_j}{\partial x_j} = f_i - \frac{1}{\rho}\frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial x^2{}_i}$$

At this point u is velocity of x direction, f is external force for unit volume, P is pressure, ρ is density and μ is viscosity.

Einstein convention form Navier-Stokes equation can summarize like under formula, if represent form of vector.

$$\frac{\vartheta u}{\vartheta t} + (\mathrm{u} \cdot \nabla)\mathrm{u} = \mathrm{f} - \frac{1}{\rho}\nabla p + v\Delta u$$

In this paper use Newtonian Navier-Stokes equation form.

Newtonian Navier-Stokes equation means Navier-Stokes equation of flow which satisfied Newtonian's flow's condition $\tau = \mu \frac{du}{dy}$.

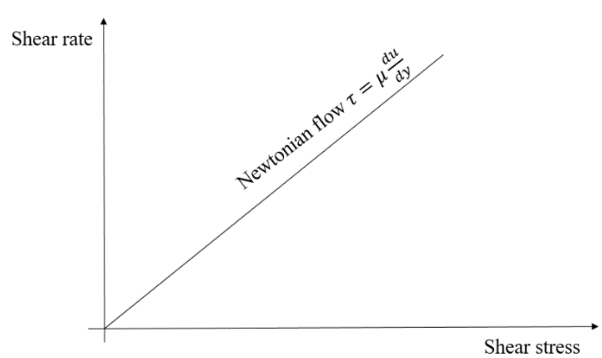


**Fig 5 Newtonian flow**

Such as Newtonian Navier-Stokes equation's component equation is lower formula.

$$\rho\frac{du}{dt}=\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=\rho g_x-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)$$

$$\rho\frac{dv}{dt}=\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=\rho g_y-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)$$

$$\rho\frac{dw}{dt}=\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=\rho g_z-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)$$

Such as component is form of final Navier-Stokes equation about Newtonian flow. Navier-Stokes equation about Newtonian flow include four unknown quantity $(\rho, u, v, w)$. So solve the Navier-Stokes equation, it need four equations. But in an instance of component equation are only three. So it needs to be alliance or solved by incompressibility continuity equation, or be removed term is considered as under the condition a constant number.

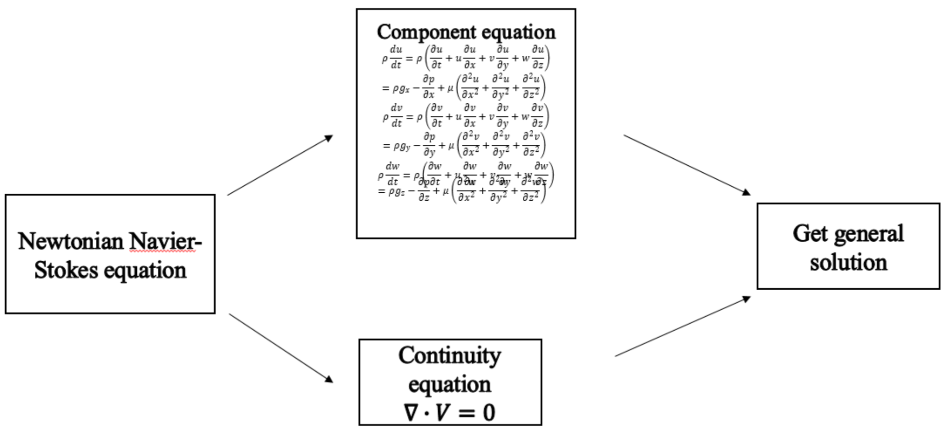


**Fig 6 process of extract Navier-Stokes equation's general solution**

### 2.2 Governed Equation in Research

To select blood vessel model we decide condition of model. A fluid of flow model's characteristics that Newtonian fluid, incompressibility, two dimension, and steady flow. Also temperature steady to 'T'. This model's characteristic is more similar water rather than blood because we need to make model more simply.
So assume fluid's viscosity coefficient is 1.002 and temperature is 20 ℃. For model's flow become second dimension flow, z axis terms are gone and also pressure of y axis direction are gone too.
Finally deduct equation through these terms.

$$\rho g-\frac{\partial p}{\partial x}+\mu\left(\nabla^2(V)\right)=\rho\frac{dV}{dt}$$

This equation is model's governed equation in our research.

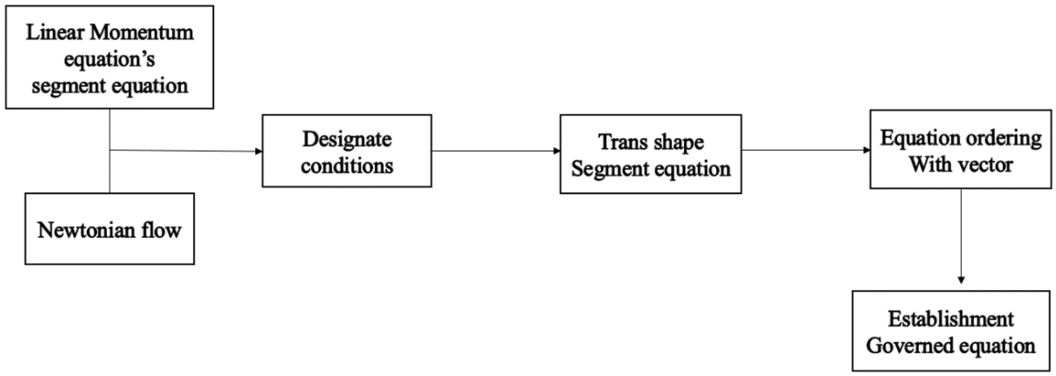


**Fig 7 process of establish governed equation.**

## 3. Interpretation about CFD (Computational Fluid Dynamics)

### 3.1 Produce and Verify the Code

In this paper programming the code about duct flow that cholesterol exist and modification code that detail conditions through similarity law are designated. Among them, in this section analyze duct flow that designated input and output velocity and made cholesterol form through reset mesh coordinate. So made code that analyze flow of model. After made code, verify code to use commercial CFD program. First Fig 7 is part of duct flow code. In this code x, y axis velocity, pressure, relationships of mesh were previously set up.

```
#include <stdio.h>
#include <stdlib.h>
#include <time.h>
#include <math.h>
#define grid 101
#define a 30
#define b 10
#define D 100
#define Re 100

int main(void)
{
    //mesh variable
    double u[grid][grid + 1], un[grid][grid + 1], uc[grid][grid];
    double v[grid + 1][grid], vn[grid + 1][grid], vc[grid][grid];
    double p[grid + 1][grid + 1], pn[grid + 1][grid + 1], pc[grid][grid];
    double m[grid + 1][grid + 1];

    //array variable
    double dy[grid], Hdy[grid], Hdy2[grid];

    //general variable
    int i, j, step;
    double dx, dt, tau, delta, error;
    step = 1;
    dt = 0.001;
    delta = 4.5;
    error = 1.0;
```

**Fig 7 code about variable**

```
        // Boundary conditions
        for (j = 1; j <= (grid - 1); j++)
        {
            un[0][j] = 1.0;
            un[grid - 1][j] = un[grid - 2][j
        }

        for (i = 0; i <= (grid - 1); i++)
        {
            un[i][0] = un[i][1];
            un[i][grid] = 0.0;
            un[i][grid - 1] = 0.0;
        }
```

**Fig 8 code about boundary condition**

Equations below are continuity equations about velocity, pressure that used in code.

$$\frac{\partial \mathrm{u}}{\partial \mathrm{t}} + \frac{\partial uu}{\partial x} + \frac{\partial uv}{\partial x} = -\frac{\partial p}{\partial x} + \vartheta \nabla^2 u$$

$$\frac{\partial \mathrm{u}}{\partial \mathrm{t}} = \frac{u^{new} - u}{dt}$$

$$\frac{\partial \mathrm{uv}}{\partial \mathrm{t}} = \frac{\left(\frac{\mathrm{u_p} + \mathrm{u_N}}{2}\right)\left(\frac{\mathrm{v_3} + \mathrm{v_4}}{2}\right) - \left(\frac{\mathrm{u_p} + \mathrm{u_S}}{2}\right)\left(\frac{v_1 + \mathrm{v_2}}{2}\right)}{\mathrm{dv}}$$

$$\frac{\partial P}{\partial x} = \frac{p_e + P_w}{2}$$

$$\vartheta \nabla^2 u = \vartheta \frac{u_E - 2u_p + u_w}{\mathrm{dx}^2} + \vartheta \frac{u_N - 2u_p + u_S}{\mathrm{dy}^2}$$

Visualization code through post processing program named 'Tecplot' is Fig 9

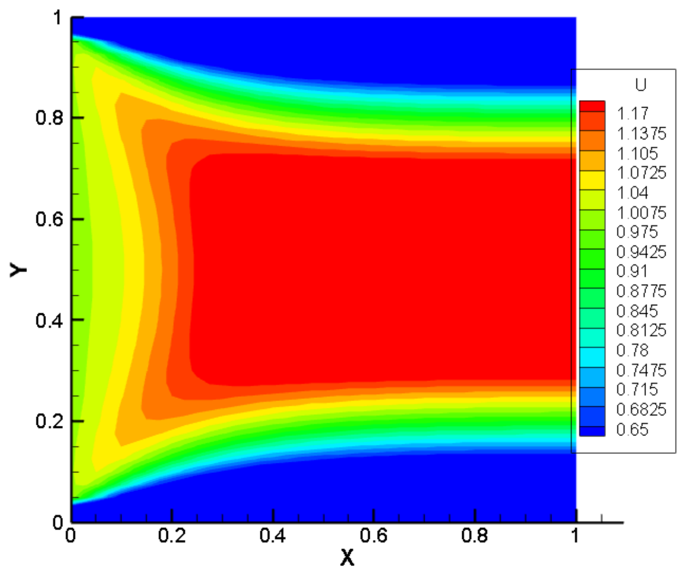


**Fig 9 Simulation result with boundary conditions from this research**

To verify the result, analyze model at program 'Tecplot'.

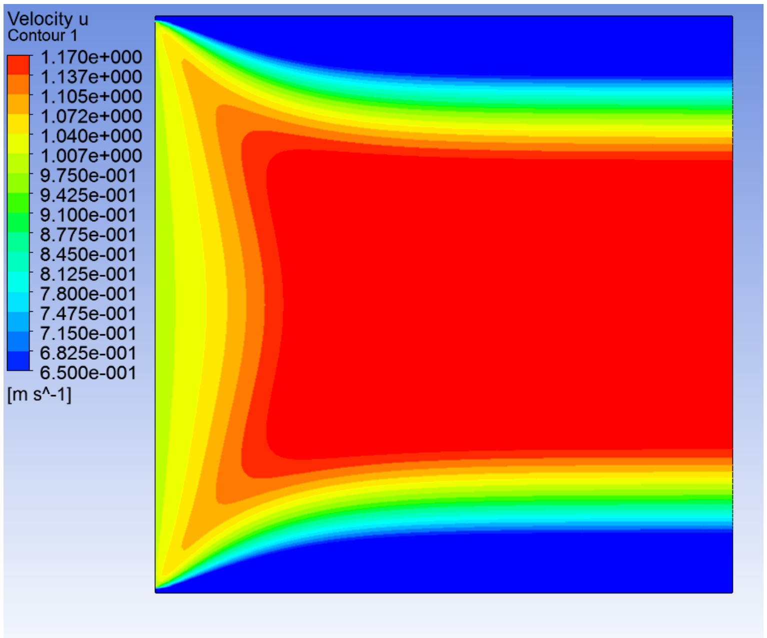


**Fig 10 Result of commercial CFD**

Finally we can realize code is correct through information that Fig 9 have similar form with Fig 10.
If cholesterol is oval, elliptic equation is

$$\frac{x^2}{a^2} + \frac{y^2}{b^2} = 1$$

At this equation 'a' is minor axis and 'b' is major axis. So it can transform to equation about y is

$$y = b\sqrt{1 - \frac{x^2}{a^2}}$$

And for ahead width of oval 's' is

$$S = 2b\sqrt{1 - \frac{x^2}{a^2}}$$

So subtract width of oval at height of control volume is height of new control volume about each x position. If height of basic control volume is D,

$$l = D - 2b\sqrt{1 - \frac{x^2}{a^2}}$$

And divide new control volume's height for number of y direction mesh is each mesh's height. So decide number of j direction mesh to $\text{grid}_j$.

$$dy = \frac{D - 2b\sqrt{1 - \frac{x^2}{a^2}}}{grid_i}$$

But number of line direction mesh is 'i'. so dy can be change to dy[i] that array function about i.

$$dy[i] = \frac{D - 2b\sqrt{1 - \frac{i^2}{a^2}}}{grid_i}$$

Oval equation is equation that centered the starting point. But number of mesh is $\text{grid}_j$. And it increase one by one start from starting point.
So change $i^2$ to $\left(i - \frac{\text{grid}_i - 1}{2}\right)^2$

$$dy[i] = \frac{D - 2b\sqrt{1 - \frac{\left(i - \frac{gird_i - 1}{2}\right)^2}{a^2}}}{grid_i}$$

```
//general variable
int i, j, step;
double dx, dt, tau, delta, error;
step = 1;
dt = 0.001;
delta = 4.5;
error = 1.0;

//generate mesh
    dx = 1.0 / (grid - 1);

for (i = 0; i <= (grid - 1); i++)
{
    dy[i] = 1.0 / (grid - 1);
}


for (int i = (grid - 1) / 2 - a + 1; i <= (grid - 1) / 2 + a - 1; i++)
{
    Hdy[i] = pow(i - ((grid - 1) / 2), 2.0);
    Hdy2[i] = sqrt((double)(pow(a, 2.0) - Hdy[i]));
    dy[i] = (a*D/2 - (b * Hdy2[i])) / (100 * a * (grid - 1));

}


//print dy[i]
for (i = 0; i <= (grid - 1); i++)
{
    printf("%.6lf₩n", dy[i]);
}
```

**Fig 11 Code of designate cholesterol form**

At this code result value is change caused by define value of major axis ‘a’, minor axis ‘b’ and length of control volume D. Again this time verify code through program ‘fluent’. And determine code is correct.

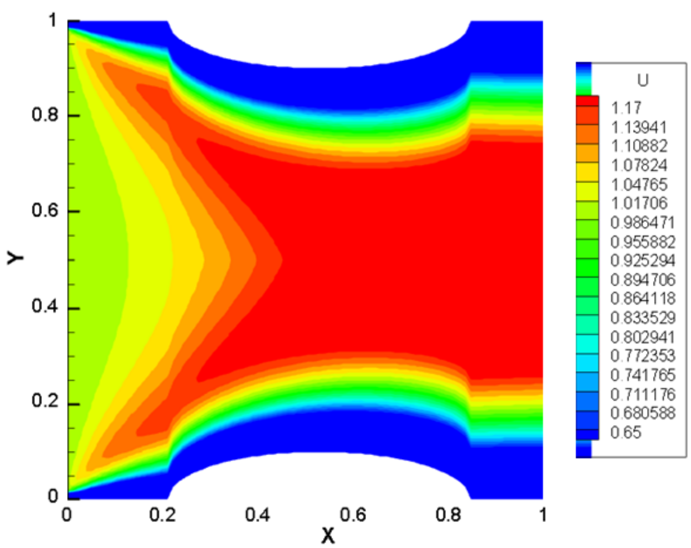


**Fig 12 Plot of fluid flow from the research**

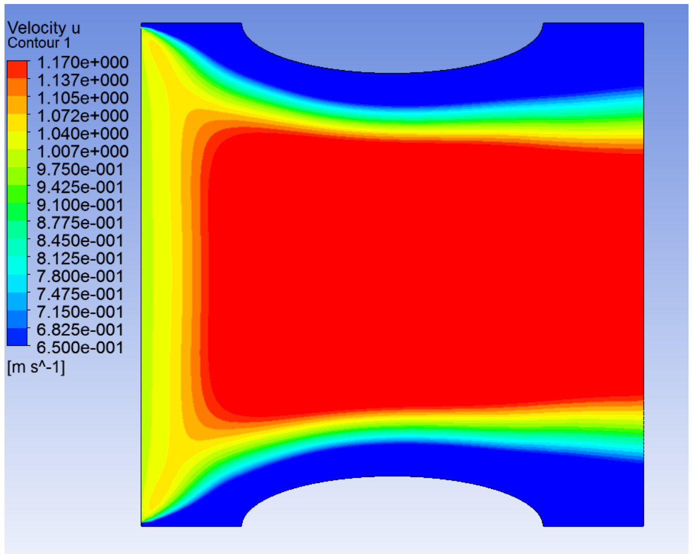


**Fig 13 Result of commercial CFD**

## 5. Conclusion and Discussion

The Navier-Stokes equation has verified its solution's existence. But it is very rare to get correct solution. So recently, CFD which develop with computer is trend to analysis flow and then get approximate solution used analysis of numerical value. In this study, analysis duct flow like cholesterol adhere at the blood vessel wall, with 2-D Navier-Stokes equation, CFD and cording. In this paper, get knowledge about discrete equation which base of CFD, mathematical and fluid dynamical knowledge. Further designate modeling Navier-Stokes equation and establish governed equation for ideal duct flow. Lastly action CFD and output result value, and get result value to use source code through C language cording

The research for analysis blood flow modeling in the same conditions with blood flow, so realization reality modeling through apply blood's properties to modeling through Law of similarity which based on study angiology. And analysis reality blood flow with various ways. And make public our source code to help researcher who research same topic. We study and analysis various model and finally we analysis traffic flow about Seoul downtown street.

## References

Adam P., "The Navier-Stokes Equations", 2004 (ftp://texmex.mit.edu/pub/emanuel/CLASS/12.340/navier-stokes(2).pdf)

Davit, M., "Large deviations for invariant measures of white-forced 2D Navier-Stokes equation" department of mathematics in cergy-pontoise university, 2016

Frank M. White, (2012) Fluid dynamics, Mc graw hill

Kim, W., study of numerical analysis for fluid flow and sediment transport processes", Yonsei University, 2000 (in Korean.)

Kim, Y., "A Fast Solver for Navier-Stokes Equations and Its Application to Real-Time Fluid Simulations", Kyungpook National University, 2005 (in Korean).

Kim, J., Sung, K., & Ryou, H., "NUMERICAL ANALYSIS ON THE BLOOD FLOW CHARACTERISTIC IN THE ARTERIOVENOUS GRAFT FOR DIFFERENT INJECTION METHOD OF BLOOD" Korean society of computational fluids engineering, v.18 no.3, pp.17-19, 2003 (in Korean).

Kim, T., Oh, S., & Yee, K., "Verification of the Open Source Code, OpenFOAM to the External Flows" Journal of the Korean Society for Aeronautical & Space Sciences, v.39 no.8, 2011년, pp.702-710, 2011 (in Korean).

Myung, H., "Unsteady 2-D Laminar Flow Simulation past a Circular Cylinder" Korean society of computational fluids engineering, no.4 no.27, pp.41 – 47, 2004 (in Korean).

Park, M., "Various application of Navier-Stokes equation and Fluid dynamics" Korean society of hazard mitigation v.14 no.2, pp.58 - 63, 2014 (in Korean).

Myung, H., (2012), CFD, Myun Un Dang, Seoul

Jang, T., "Flow Induced Vibration Analysis Using Two-Dimensional Incompressible Navier-Stokes Equations", Korea Institute of Science and Technology, 2002 (in Korean).

Jang, S., "A Numerical Study on The Characteristics of The Fluid Flow in A Backward-Facing Step Channel with A Porous Body" ,Korea Institute of Science and Technology, 1986 (in Korean).

Suh, S., & Yoo, S., "Steady Flow Analyses of Blood Analogue Fluids in the Stenosed Circular and Bifurcated Tubes", Korea-Australia Rheology Journal, v.7, no.2, pp. 150-157, 1995 (in Korean).

Kim, Hyea-Hyun. "Numerical analysis of Navier-Stokes equations", Korea Institute of Science and Technology, 1999 (in Korean)