Fluid Flow Simulations for Biomedical Applications

Samuel Scruggs,
Mentor : Kwai Wong
The University of Tennessee, Knoxville,TN

Abstract

Through Computational Fluid Dynamics (CFD) fluid flow in various geometries can be predicted. More specifically, airflow through the nose, mouth, and lungs can be simulated with various programs to help calculate and display the flow. By analyzing the results of these calculations, certain conditions applied to the geometry can be altered in order to achieve a more accurate prediction of the flow result.

Meshing

In order to get the most accurate representation of fluid flow, the mesh of the geometry is important. A mesh with more points allows for more data to be calculated and analyzed.

Mesh (b) is a better mesh because it has more points for the calculations.

a)

b)
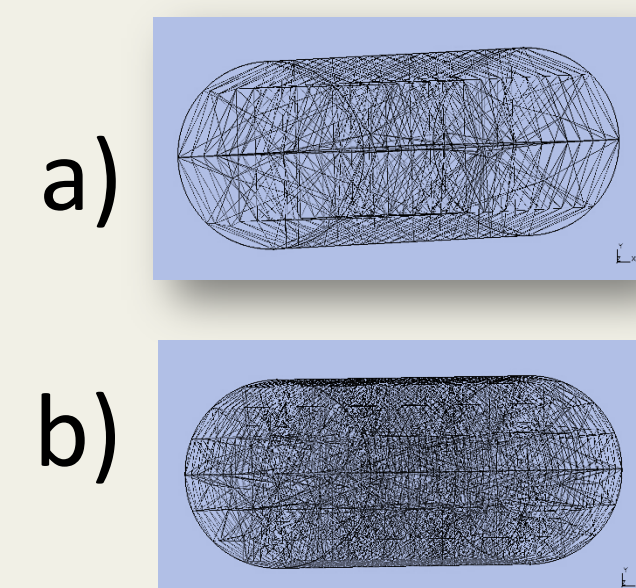
In the case of a hollow tube with water flowing through it, a more realistic result will arise from mesh (b) due to the mesh being more detailed.

The program used in this case to help create the mesh is Gmsh. Multiple other programs such a cubit and ANSYS Workbench can be used to help create geometry meshes.

Boundary Conditions

Conditions can be established for the mesh before utilizing the solver. This helps to set up what the fluid is actually doing in the geometry.
For Example: An ordinary pipe has an inlet established at one end, walls that surround the open area, and an outlet at the opposite end. Along with the inside being categorized as air, these boundary conditions are established with the mesh in order for the solver program to read.

Solver

The equations used to solve fluid flow through geometries are the Navier-Stokes Equations.

- Axisymmetric INS

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial r} + w\frac{\partial u}{\partial x} = -\frac{1}{\rho}\frac{\partial p}{\partial r} + \nu(\frac{\partial^2 u}{\partial r^2} + \frac{1}{r}\frac{\partial u}{\partial r} + \frac{\partial^2 u}{\partial x^2} - \frac{u}{r^2})$$

$$\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial r} + w\frac{\partial w}{\partial x} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu(\frac{\partial^2 w}{\partial r^2} + \frac{1}{r}\frac{\partial w}{\partial r} + \frac{\partial^2 w}{\partial x^2})$$

$$\frac{1}{r}\frac{\partial}{\partial r}(ru) + \frac{\partial w}{\partial x} = 0,$$

These various equations are used by solver programs to calculate the result of the fluid flow.
For certain situations, there are different initial conditions that are made. Velocity of the flow can change, density of the fluid, and even the material of the geometry itself are just a few of the possible variables that could change the outcome.

These equations will also be used when it comes to solving fluid flow throughout the mouth, nose, and lungs.

Formatting

The geometry's file must be formatted properly in order for each program to read it and perform its task:
1) Geometry and Mesh created using the program Gmsh. Boundary conditions are defined in the mesh
2) Using the gmshToFoam command the mesh file can be converted from Gmsh to OpenFoam
3) OpenFoam can then be used to perform specific calculations for the fluid flow with different initial conditions
4) The result of these calculations are then read in ParaView to be analyzed
5) Through benchmark tests and previous knowledge of fluid flow, the reasonability of the results can be determined by viewing the results in ParaView
6) Mistakes are then corrected by refining the mesh and boundary conditions and then running the geometry back through the different programs

Acknowledgements

This project is supported by the National Science Foundation

Post-Processing

There are several post-processing units used to view the results of the solver and in this case the program ParaView was used.
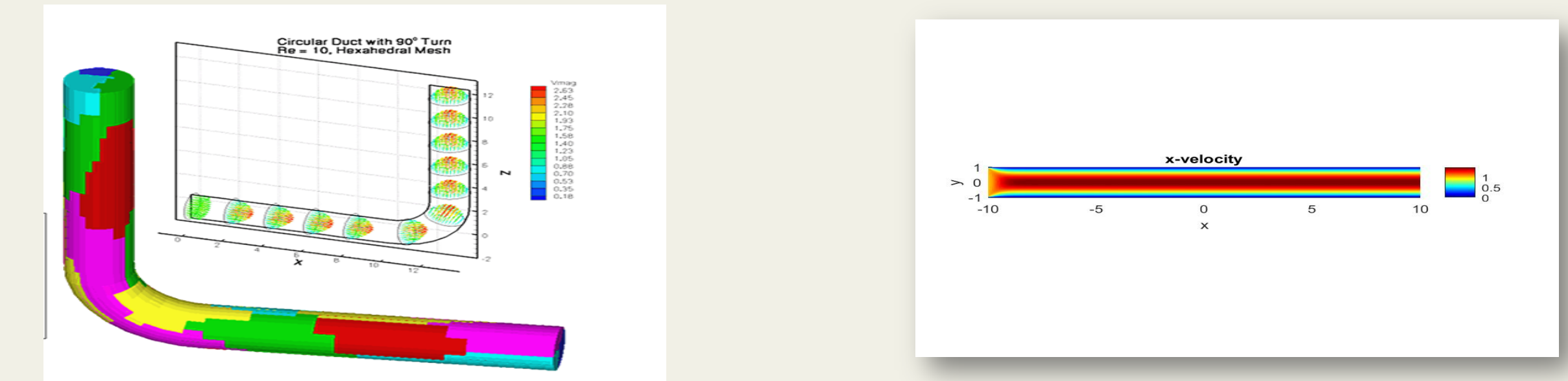
In the photo above, the result of water flowing through a pipe is shown. The effects of friction from the walls and the speed of the water at the beginning is shown through the different colors.

The same situation can be made when simulating particle flow through the mouth, nose, and lungs. Taking from the benchmark results of fluid flow, this experience can be applied to the results from future tests.
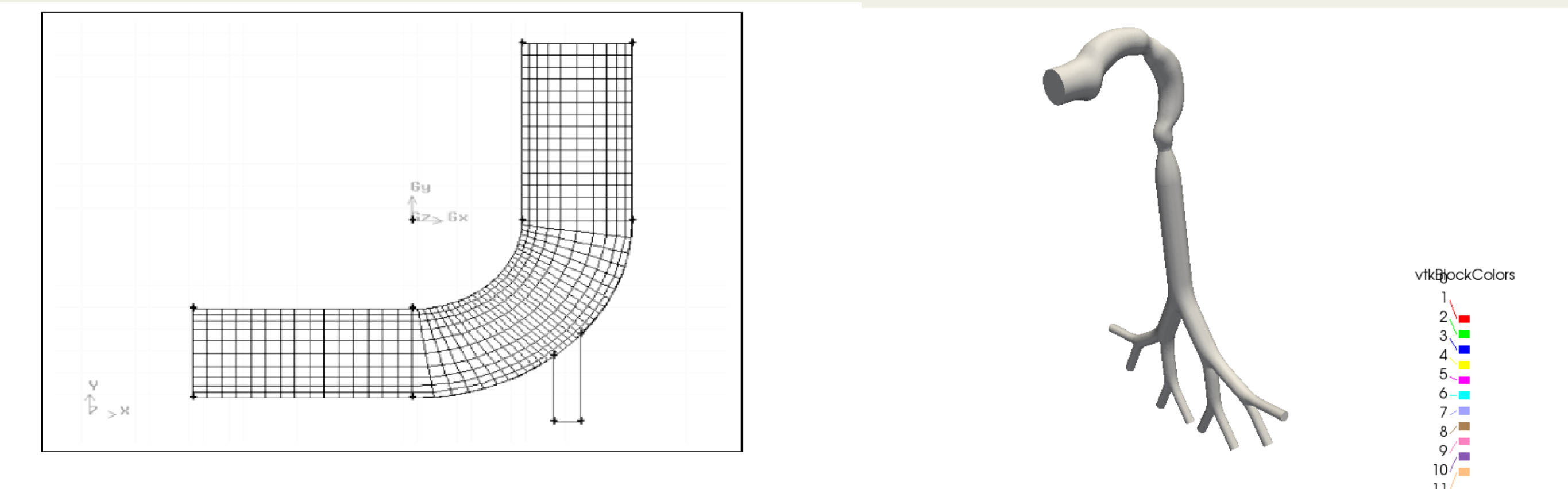
Future Research

The research to continue involves fluid flow through the mouth, nose, and lungs.
Particle flow can be analyzed using these same programs in order to this fluid flow through the body.
The effects of the injection of these particles can also be analyzed through these various programs.
In the same way that fluid flow is analyzed through a tube, these methods can be used to help determine the flow of these particles in the body.

References

www.sourceforge.net
users.ugent.be/~mvbelleg/flug-12-0.pdf
https://cubit.sandia.gov
www.openforam.org
www.paraview.org
gmsh.info