Simulations of Flow past a Complex Object in a Numerical Wind Tunnel using Direct Forcing Immersed Boundary Method

Areli Diaz, Aerospace Engineering
Mentor: Mohamed Houssem Kasbaoui, Assistant Professor
School for Engineering of Matter, Transport & Energy

How can the direct forcing immersed boundary method be applied to model the flow over a complicated geometry in a numerical wind tunnel?

Abstract

This project demonstrated the capabilities of the direct forcing Immersed Boundary method. This was done by simulating the flow past a complex object in a numerical wind tunnel using a computer graphic software called VisIt.

Background

The Immersed Boundary methods were first introduced by Charles S. Peskin in 1972 to simulate the mechanics of the heart and blood flow because no other grid systems properly conformed to the complicated geometry of the heart. Immersed Boundary Methods are models and simulations used to show interactions between a fluid and object by:

- Using Navier-Stokes equations to communicate a Eulerian grid (fluid grid) with a Lagrangian frame.[1]
- Simplifying grid generation
- Not requiring special conditions to impose along the immersed boundary[2] allowing this method to be used to study a greater range of problems.

Research Methods

- Performed a literature review to understand the theory behind the direct forcing Immersed Boundary method.
- Developed a workflow using the direct forcing Immersed Boundary method to generate a mesh for simulations of flow past a complex object in a numerical wind tunnel.
- Ran NGA research code and gathered data.
- Visualized the data by using an STL file, performing 3D rendering, and simulating the flow.

Results

A high-performance computing environment, ASU Agave Cluster, was used to run an in-house code called NGA. This code gathered data of the flow past the complex object in the numerical wind tunnel. The complex object used for the analysis between the fluid and complex object was an STL file of the Millennium Falcon starship from the Star Wars franchise. The Immersed Boundary grid was applied to the starship as shown in Figure 1 and Figure 2.

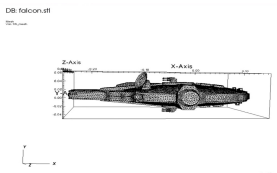

Figure 1

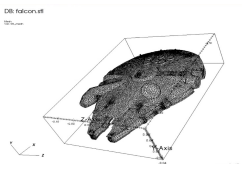

Figure 2

Using the data from computationally simulating the flow past the Millennium Falcon, the following images (Figure 3) were rendered using VisIt illustrating the magnitude of velocity at different points in time.

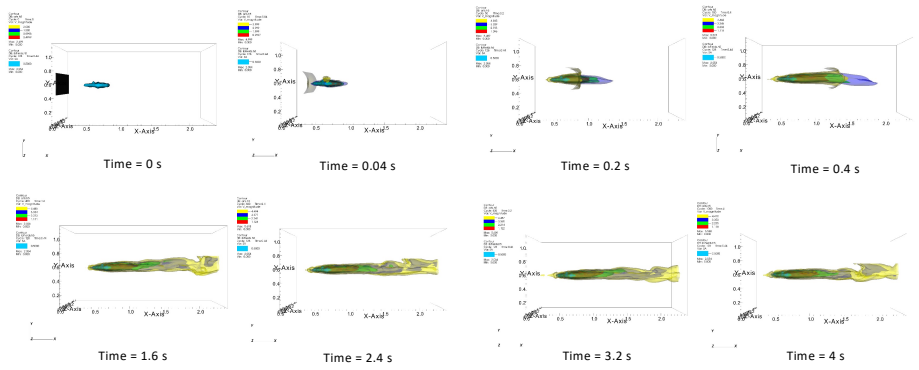

Figure 3

Conclusion

The data collected using the NGA code in the Agave Cluster high-performance environment and the images rendered from VisIt demonstrated:

- The potential of using the direct forcing Immersed Boundary method
- More data points are needed for a better rendering image.
- New rendering technique or software needed for higher quality images.

References

1. Mittal, Rajat, and Gianluca Iaccarino. "Immersed Boundary Methods." Annual Review of Fluid Mechanics37, no. 1 (2005): 239-61. doi:10.1146/annurev.fluid.37.061903.175743.
2. Peskin, Charles S. "Flow Patterns around Heart Valves: A Numerical Method." Journal of Computational Physics10, no. 2 (1972): 252-71. doi:10.1016/0021-9991(72)90065-4.

Acknowledgements

A special thank you to my mentor, Mohammed Houssem Kasbaoui, for guidance throughout this project. Also, a special thank you to Himanshu Dave for his help and guidance.

Contact Information

Areli Diaz
Email: adiaz67@asu.edu

FURI

Ira A. Fulton Schools of Engineering
Arizona State University