FLOW AROUND A BLUFF-BODY IN A CHANNEL FLOW

Andaç, Taha

1. INTRODUCTION

Focus this project is on utilizing CFD to explore the problems of fluid flow around a bluff-body in a channel. The project involves modifying the provided projection method code for a fluid flow over a channel using the given boundary conditions and the fluid properties.

The primary goal is to include addressing challenges related to force determination and temperature field analysis. The project aims to provide valuable insights into fluid-structure interactions and heat transfer processes in confined geometries. Through a holistic approach that combines theoretical foundations, numerical methodologies, and visualization techniques, the endeavor aims to advance our understanding of complex fluid dynamics—a crucial element in the design and optimization of diverse engineering systems.

2. METHODOLOGY

For computational domain creating a staggered grid was used to simulate the flow. In the grid, pressure nodes were stored in the center of the grid squares, u velocities was stored at the right edge of the squares and the v velocities was stored at the top edge of the squares as demonstrated on Fig. 1.

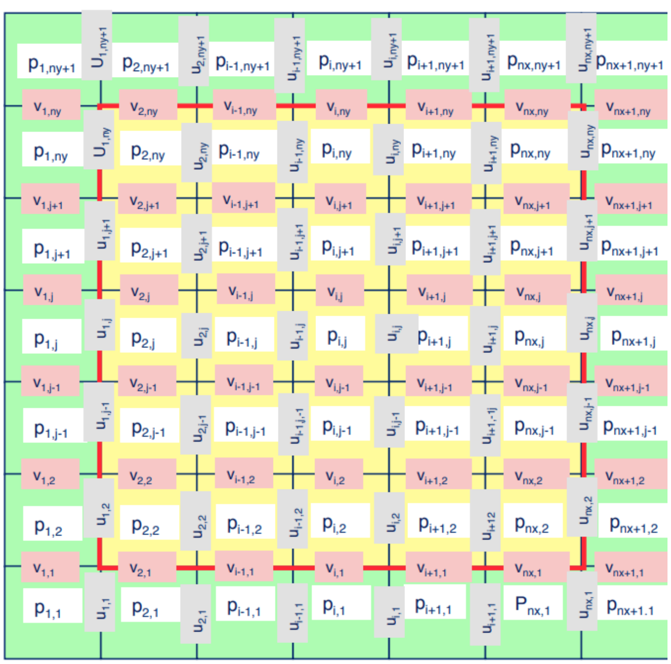

Figure 1. Staggered grid configuration.

For momentum calculations central space forward time configuration was used with the Taylor series expansions for v and u velocities individually. For P values using the convesvation of mass and velocities pressure solver was created which changes with the computation domain boundaries. After that for grid convergence I compared flow of a empty tube simulation with the analytical solution of the flow and obtained grid convergence.

3. RESULTS AND DISCUSSION

Analytical solution for the empty tube flows I compared my grid convergence for the numerical method. As seen on the Fig. 2 my computational domain creates a solution which is very similar to the analytical one.

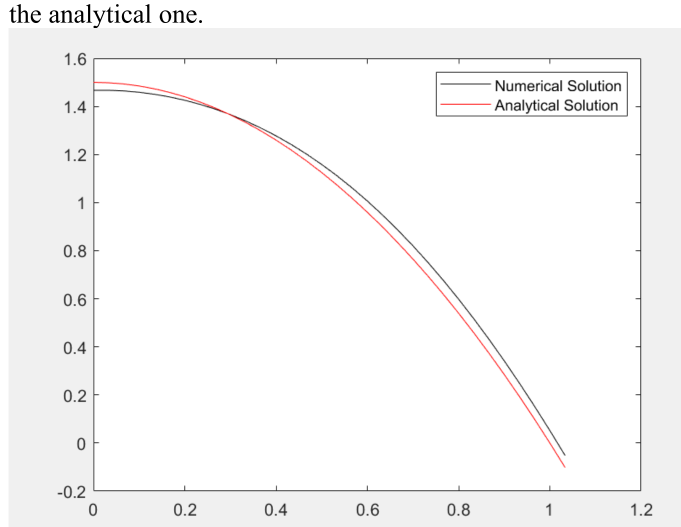

Figure 2. Analytical Solution vs Numerical Solution

After most efficient computational domain for the problem was selected. In the implementation of tube with blockage some boundary condition problems were encountered. For some reason given application of given boundary condition caused solutions to diverge. This is most probably a user error. With trial an error a boundary was applied which was very close the required one. Due to those solutions of the problem is not totally correct, however this was the only way to obtain some converged solutions. Solutions are displayed on the figures below.

Solutions for a 120-30 grid

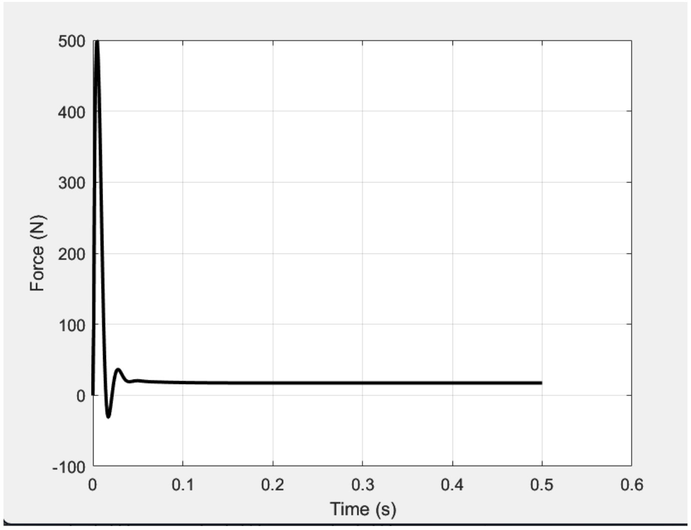

Figure 3. Force vs Time

Total force applied on the block converges into 16N eventually as seen on Fig. 3.

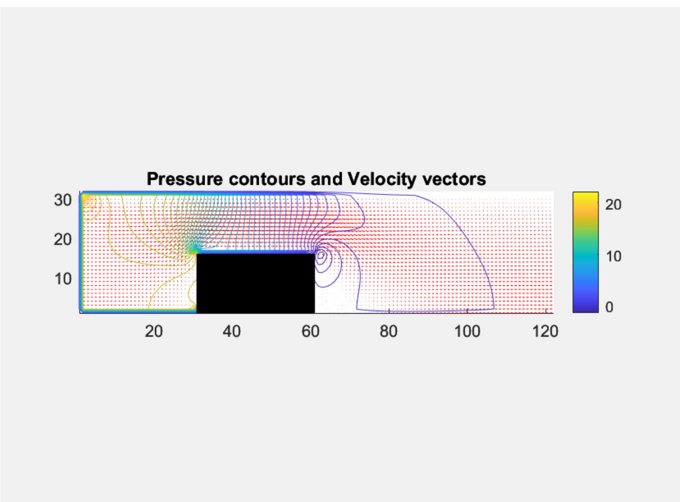

Figure 4. Pressure and Velocity solutions

As mentioned above due to the broken boundary conditions solutions are not correct at the edges of the block as seen on Fig. 4.

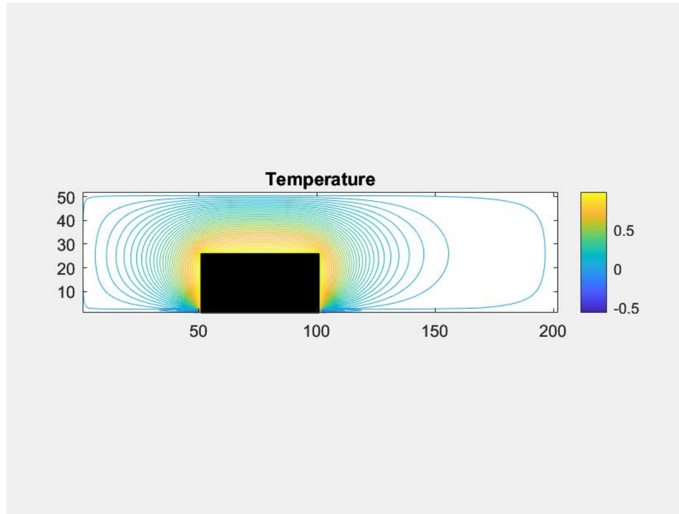

Figure 5. Temperature Solution

Temperature solutions seem reasonable since only drag causes to temperature increase as demonstrated on Fig. 5.

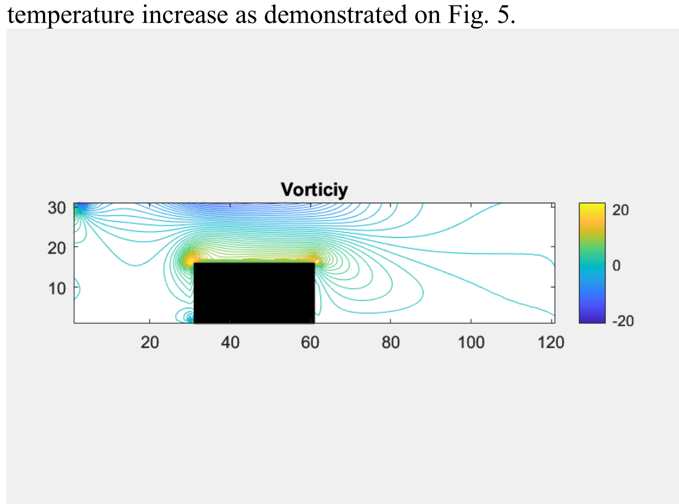

Figure 6. Vorticity Solution

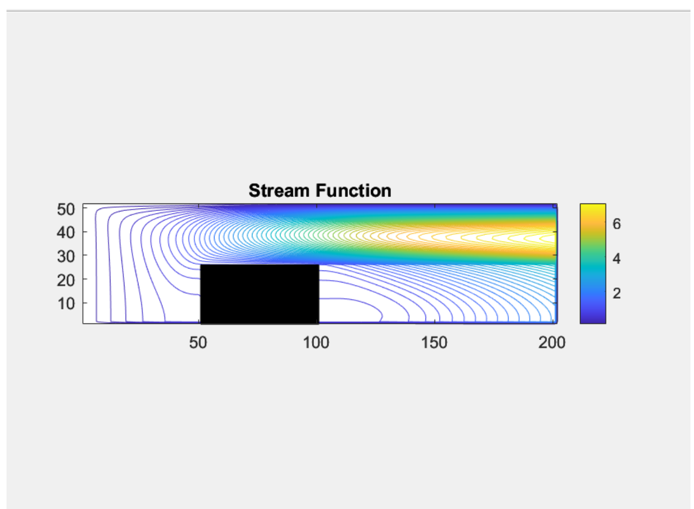

Figure 7. Stream Function Solution

Solutions for a 200-50 grid

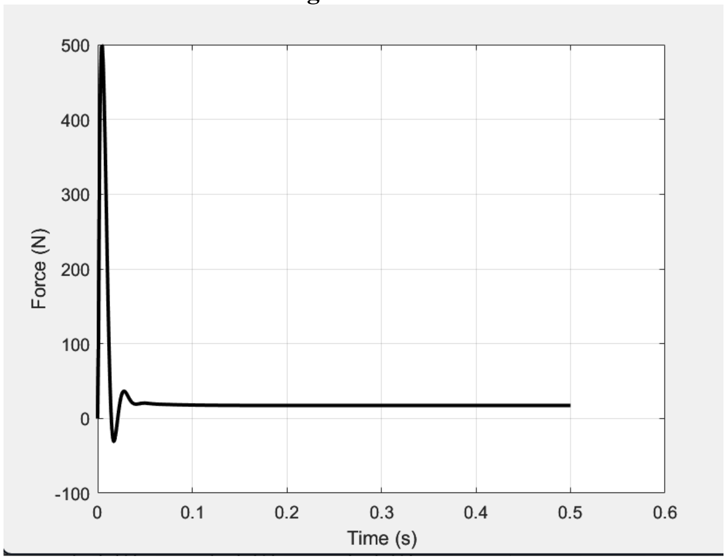

Figure 8. Force vs Time

Again, for a 200-50 force converges into 16N, proving the grid convergence.

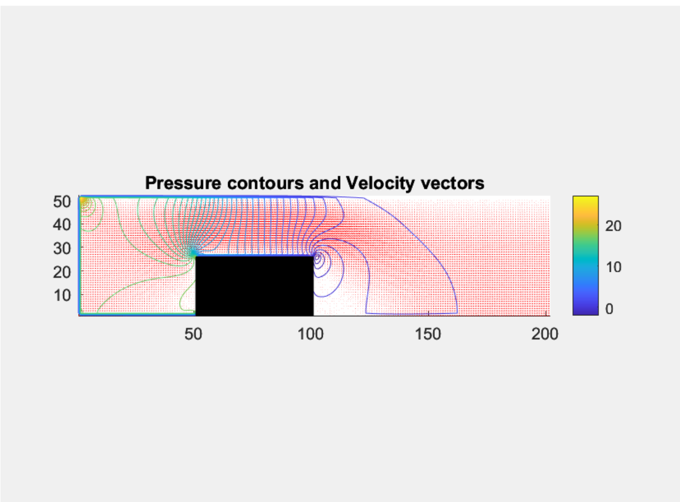

Figure 9. Pressure and Velocity solutions

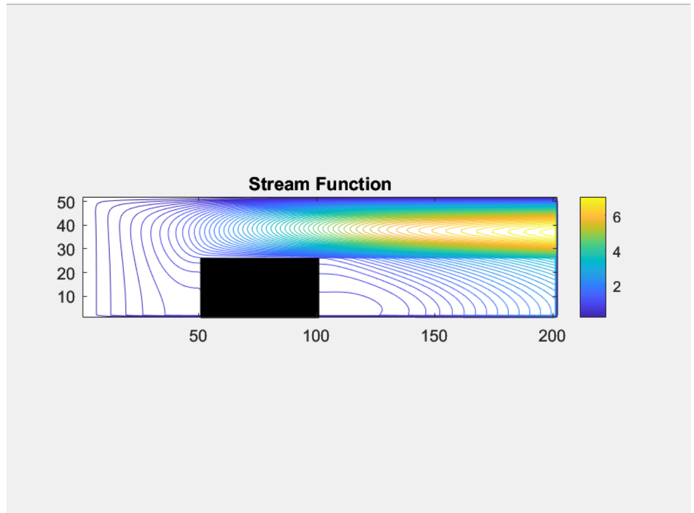

Figure 10. Stream Function Solution

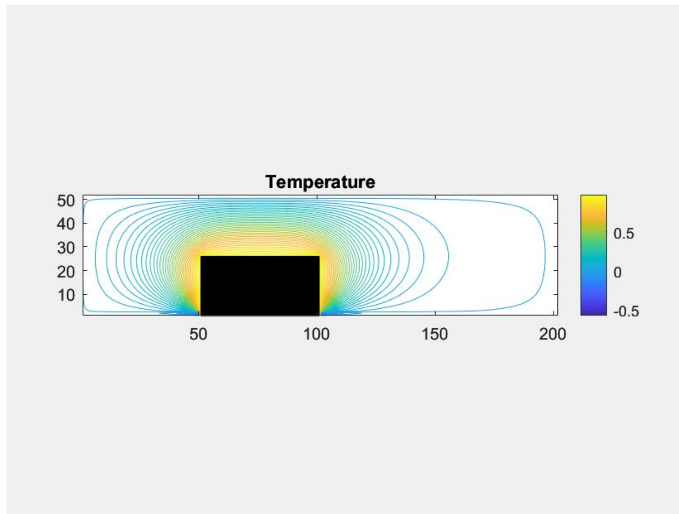

Figure 11. Temperature Solution

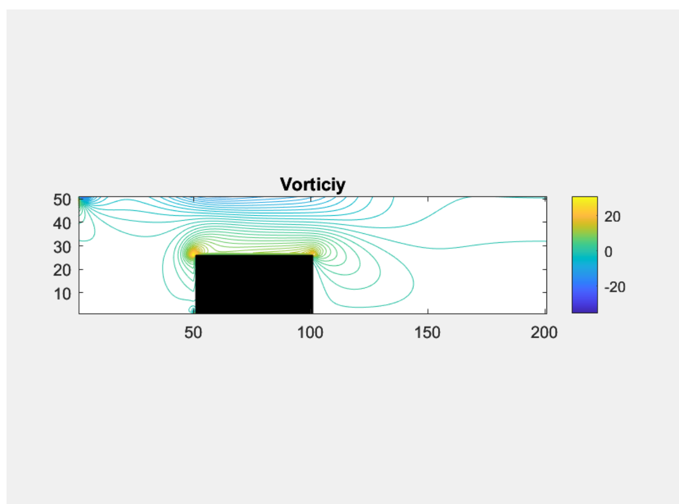

Figure 12. Vorticity Solution.

4. CONCLUSION

In a conclusion, flow around a bluff body in a channel flow was simulated with small errors here and there. Although the solution method is little problematic, different computational domains give the same solution which proves that solution is convergent.