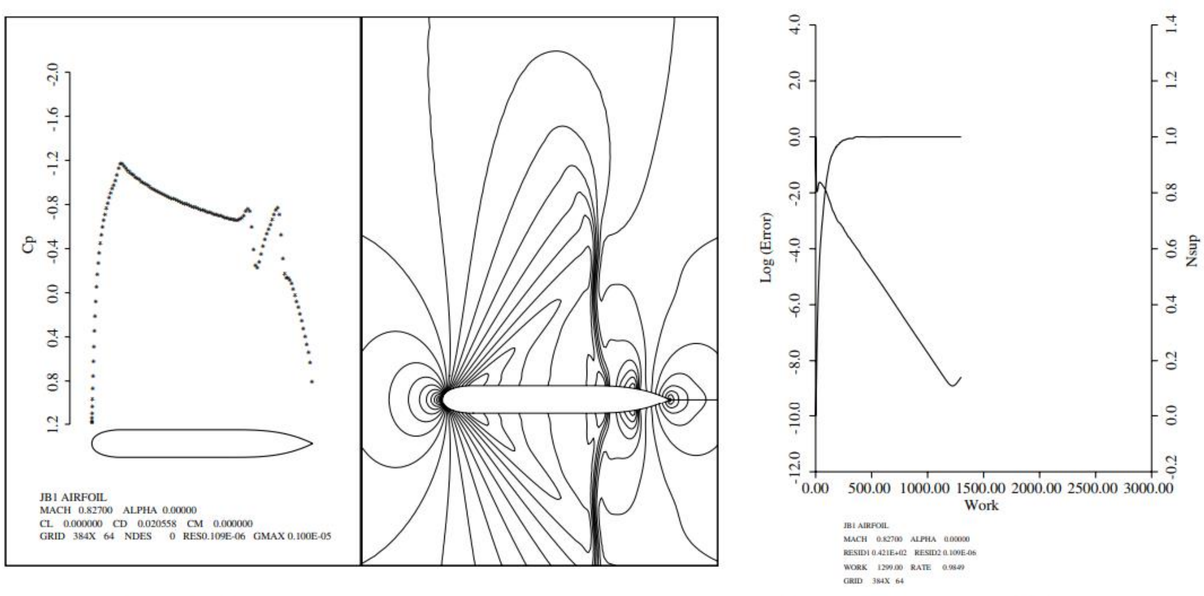
A Study on Effect of Mesh and Steadiness on Non-unique Solution of Transonic Flow

Shuheng Liu

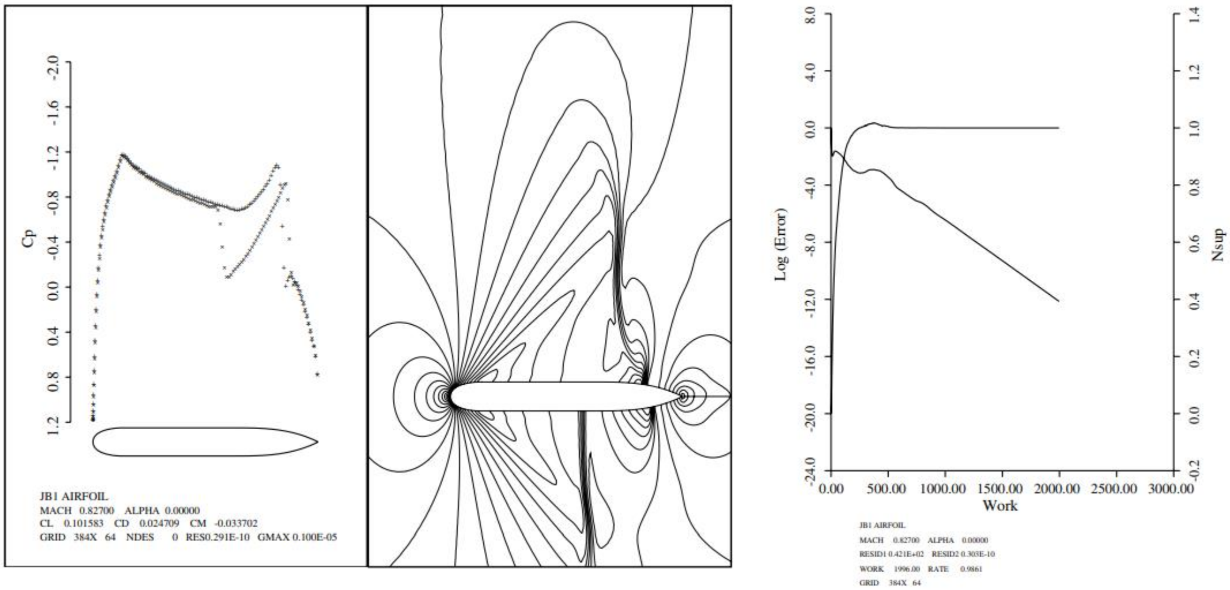
Motivation

When certain airfoils pass through transonic flow, chances are that we get non-unique solution if we carry out CFD simulation. In *Further Studies of Airfoils Supporting Non-Unique Solutions in Transonic Flow*, the authors found "four symmetric airfoils, all of which exhibit non-unique solutions in a narrow band of transonic Mach numbers. C_L-α plots of these airfoils exhibit three branches at zero angle of attach, the P-, Z- and N-branches with positive, zero and negative lift respectively. At some Mach numbers no stable Z-branch could be found. When the P-branch is continued to negative α in some cases there is transition to the Z-branch, while in other cases there is a direct transition from the P- to the N-branch." In our study, one of the four airfoils, a so-called JB1 airfoil is researched when transonic inviscid flow (M_∞=0.827) goes around it. We would like to focus on the effect of mesh and steady/ unsteady solver on the non-unique solutions. For reference results, the paper gives the Mach contour and C_L-α plot. Note the transition in C_L-α plot is that P-branch transits to N-branch by a sudden jump when α decreases to negative (Figure 1~3).



a) Z-branch Cp distribution and mach contours **b) Z-branch convergence history**

Figure 1. Reference Z-branch Mach contour and convergence history



c) P-branch Cp distribution and mach contours **d) P-branch convergence history**

Figure 2. Reference P-branch mach contour and convergence study

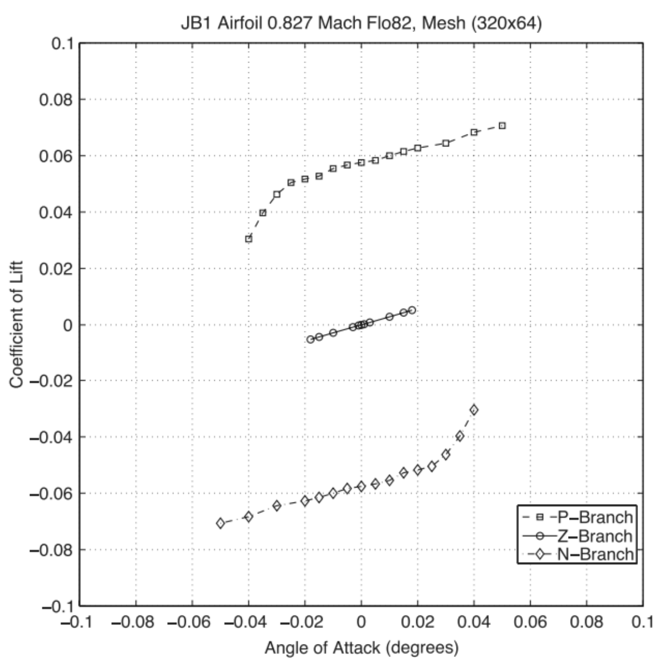


Figure 3. C_L-α plot for JB-1 airfoil, M=0.827

The authors use self-developed SYN83 code, which implements the Jameson-Schmidt-Turkel scheme on a mesh with C-topology, which contained ~300 cells in the clockwise direction and 64 cells in the normal direction. In all cases, a lifting solution at zero angle of attack was obtained by starting at angle of attack of 0.05 deg and switching to 0 deg after 500 iterations. For us, we are going to simulate the same problem by STAR-CCM+ inviscid and coupled flow solver with various meshes and steady/unsteady settings.

Procedure

1. Generate a 3D CAD model for the airfoil with boundaries

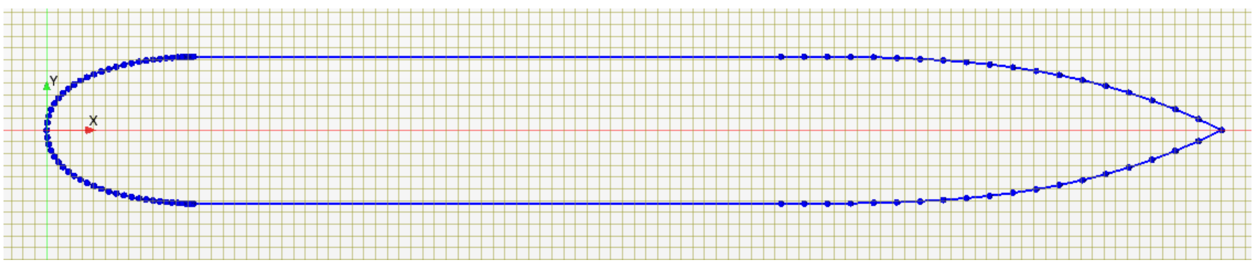
1a. Import airfoil geometry

The JB1 airfoil is given by:

$$\begin{cases} x = 0.125(3t^2 - 2t^3) \\ y = 0.0625(3t - 3t^2 + t^3) \end{cases}, 0 \le t \le 1$$

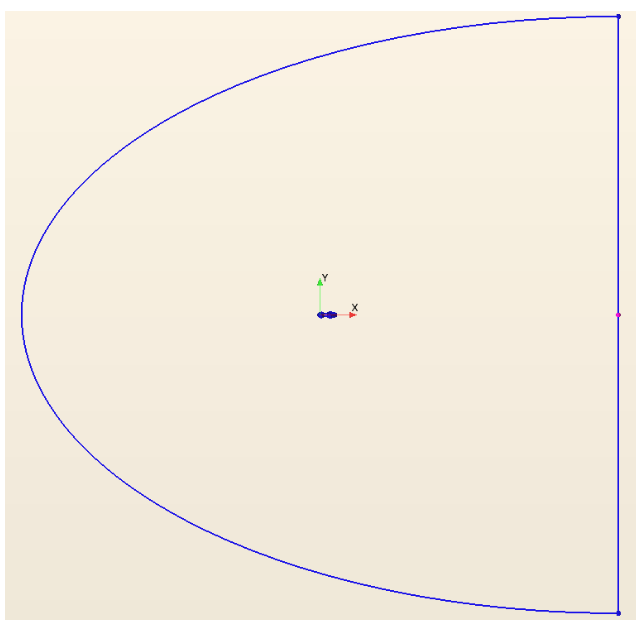
$$y = 0.0625\left\{1 - \left[1 - \left(\frac{1-x}{0.375}\right)\right]^4\right\}, 0.625 \le x \le 1$$

Import the x-y-z coordinates of JB1 airfoil by a CSV file. Here, we choose a total of 98 points to create JB-1 airfoil as follows:



1b. Create a boundary

Note the boundary should be far away from the airfoil to meet freestream condition. And here, we use a boundary like the figure below:



Here, we choose a semi-oval to surround the airfoil. The minor axis is ~320 times of the thickness, while the semi-major axis is ~40 times of the chord length. This geometry is inspired by an example of STAR-CCM+ transonic simulation (Transonic Flow: RAE2822 Airfoil)

1c. extrude to 3D geometry

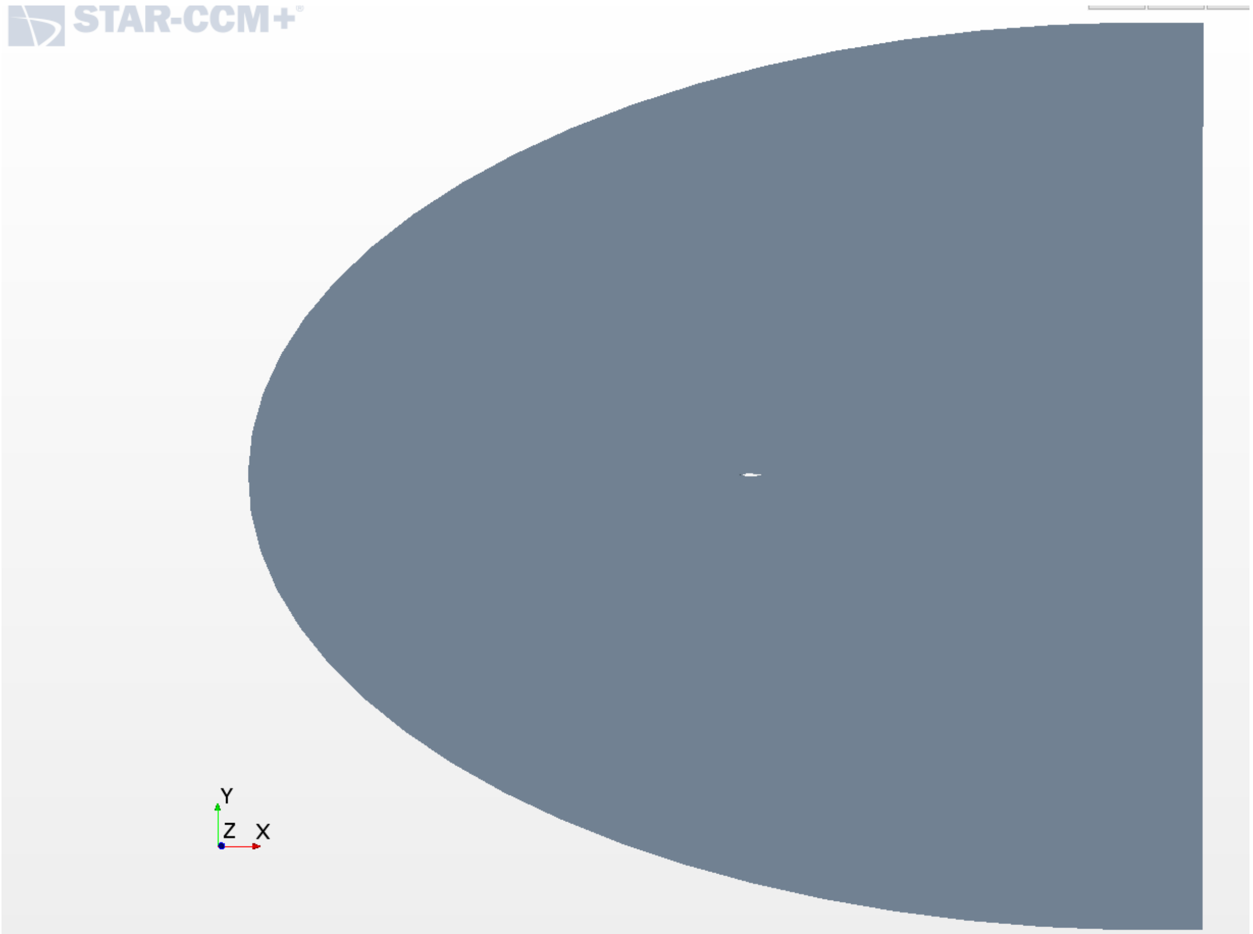


Figure 4. Whole 3D geometry of simulation

1d.

After the geometry is created, there are manipulations to convert the model to part and split the part to surfaces, for while we can assigned to regions and define boundary conditions.

They are set to be:

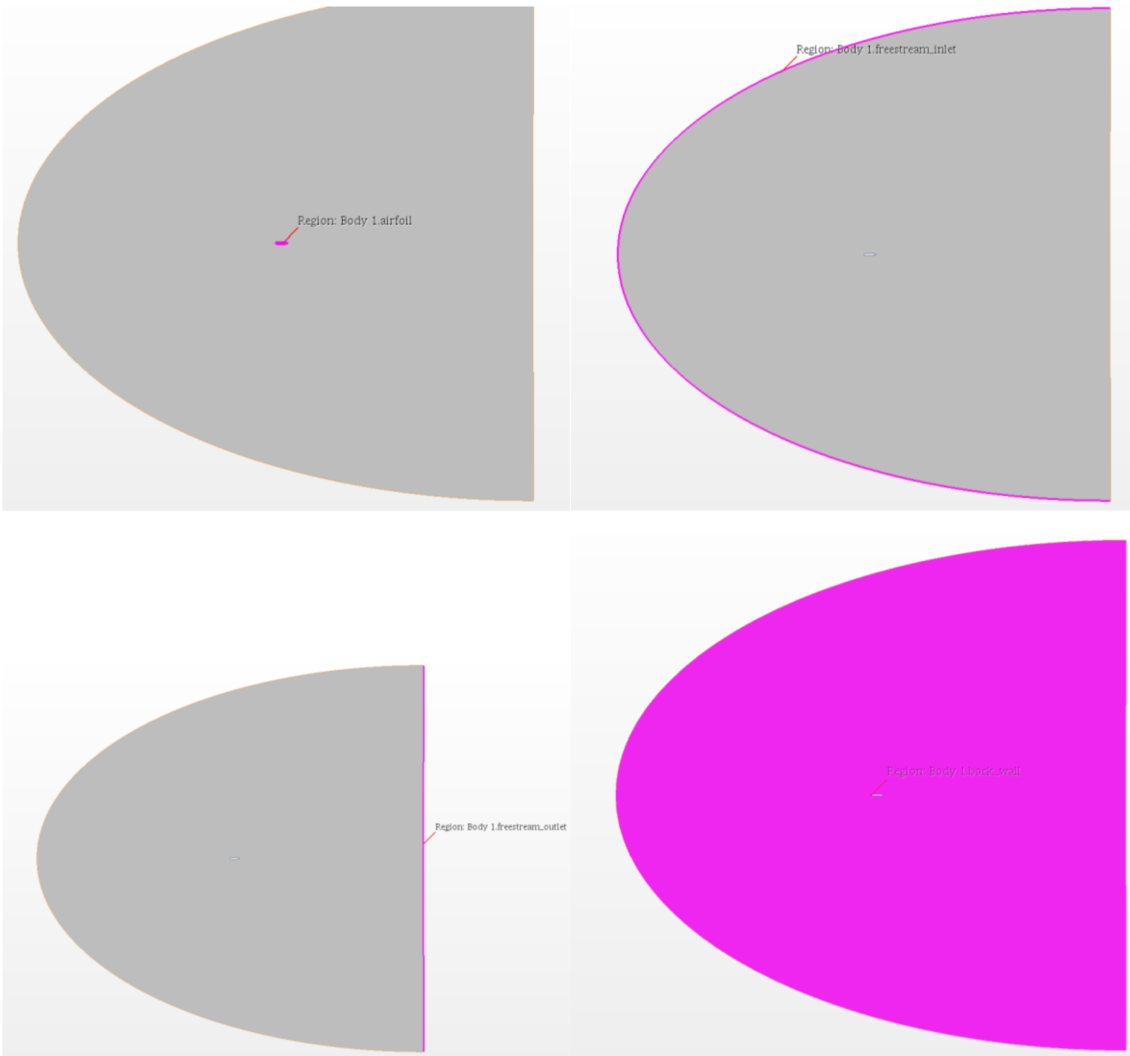
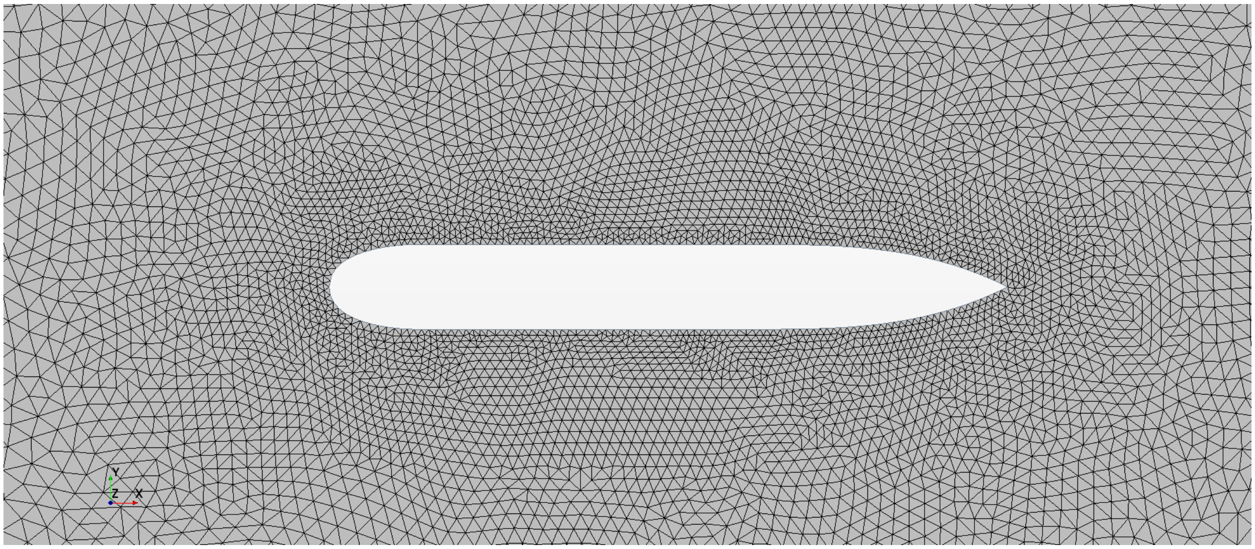


Figure 5. Boundary conditions

2. Create mesh for the 3D geometry(Triangular and Directed)

2a. Create a triangular mesh



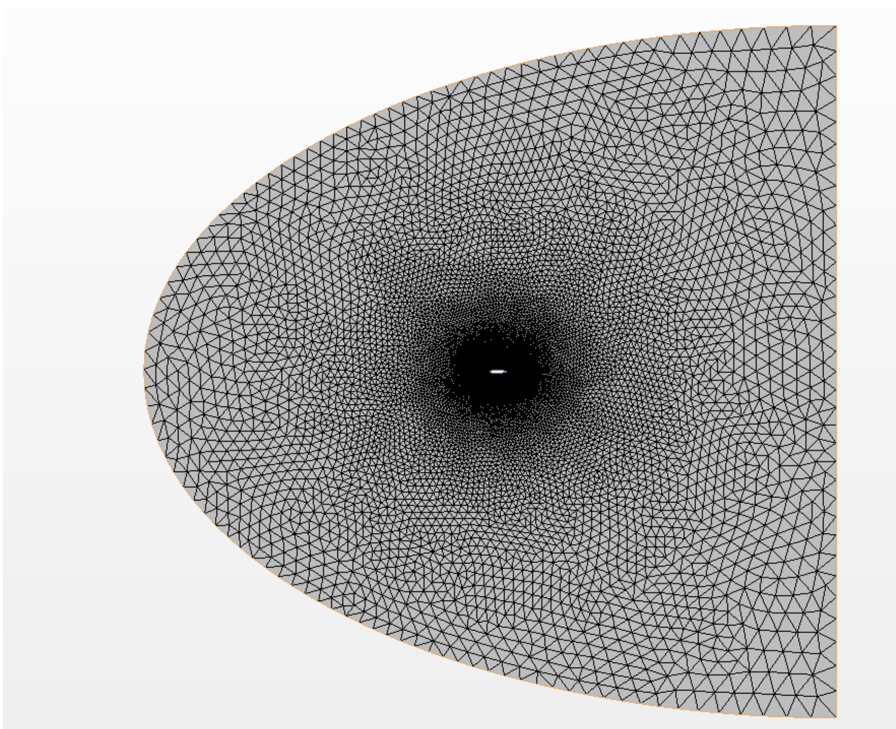


Figure 6. Triangular Mesh

This is a mesh with 29919 cells, approximate the same magnitude as the reference mesh (320*64). We call it Triangular Mesh.

2b.Create a directed Mesh

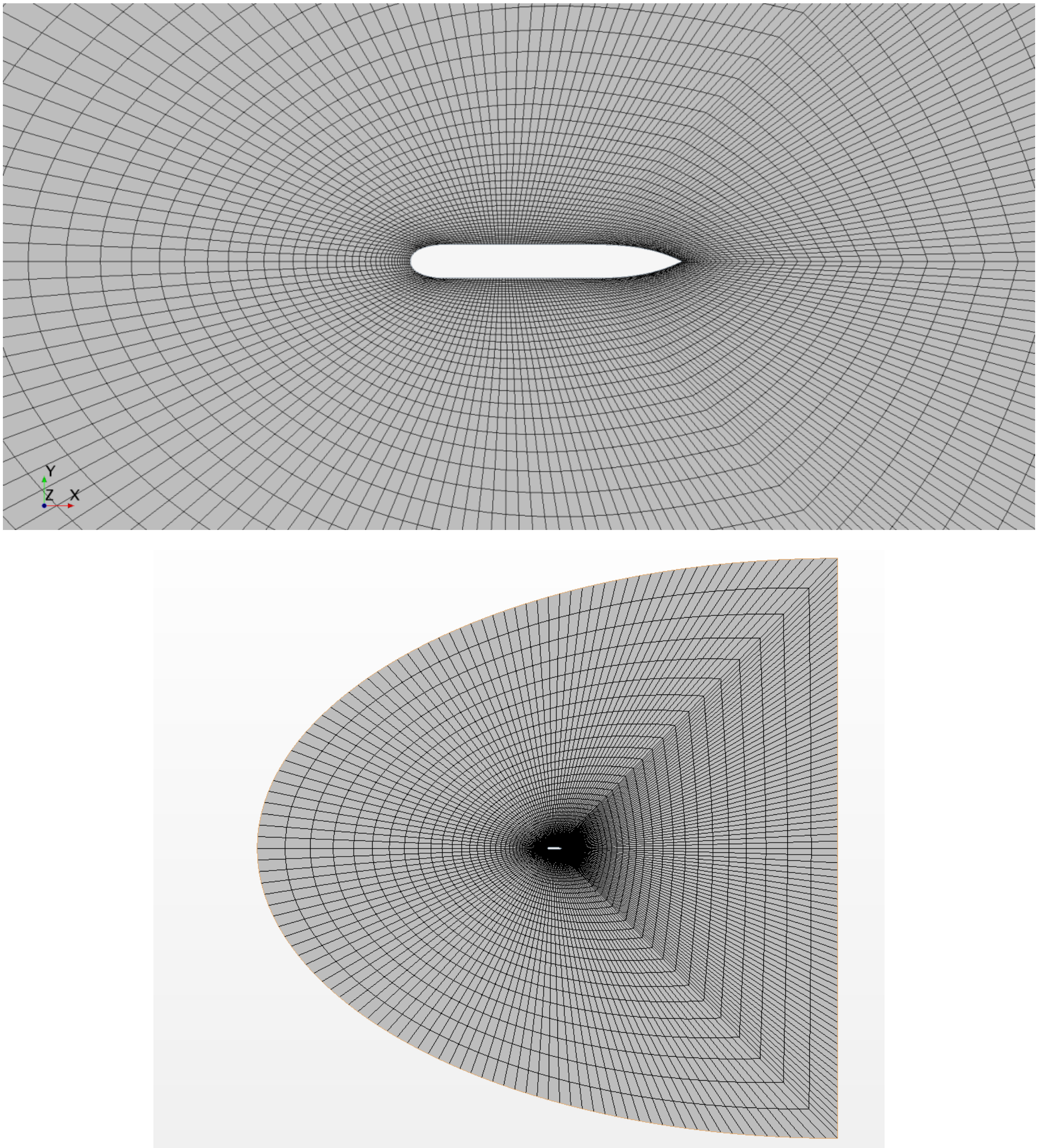


Figure 7. Coarse directed mesh

This mesh has 200*60 cells. We call it coarse directed mesh.

A same structure of mesh, which has 400*150 cells. We call it dense directed mesh.

This Mesh (Figure 8) is also 200*60, but it is locally dense, especially aiming at simulating shocks perpendicular to the periphery of the airfoil. We call it locally dense directed mesh.

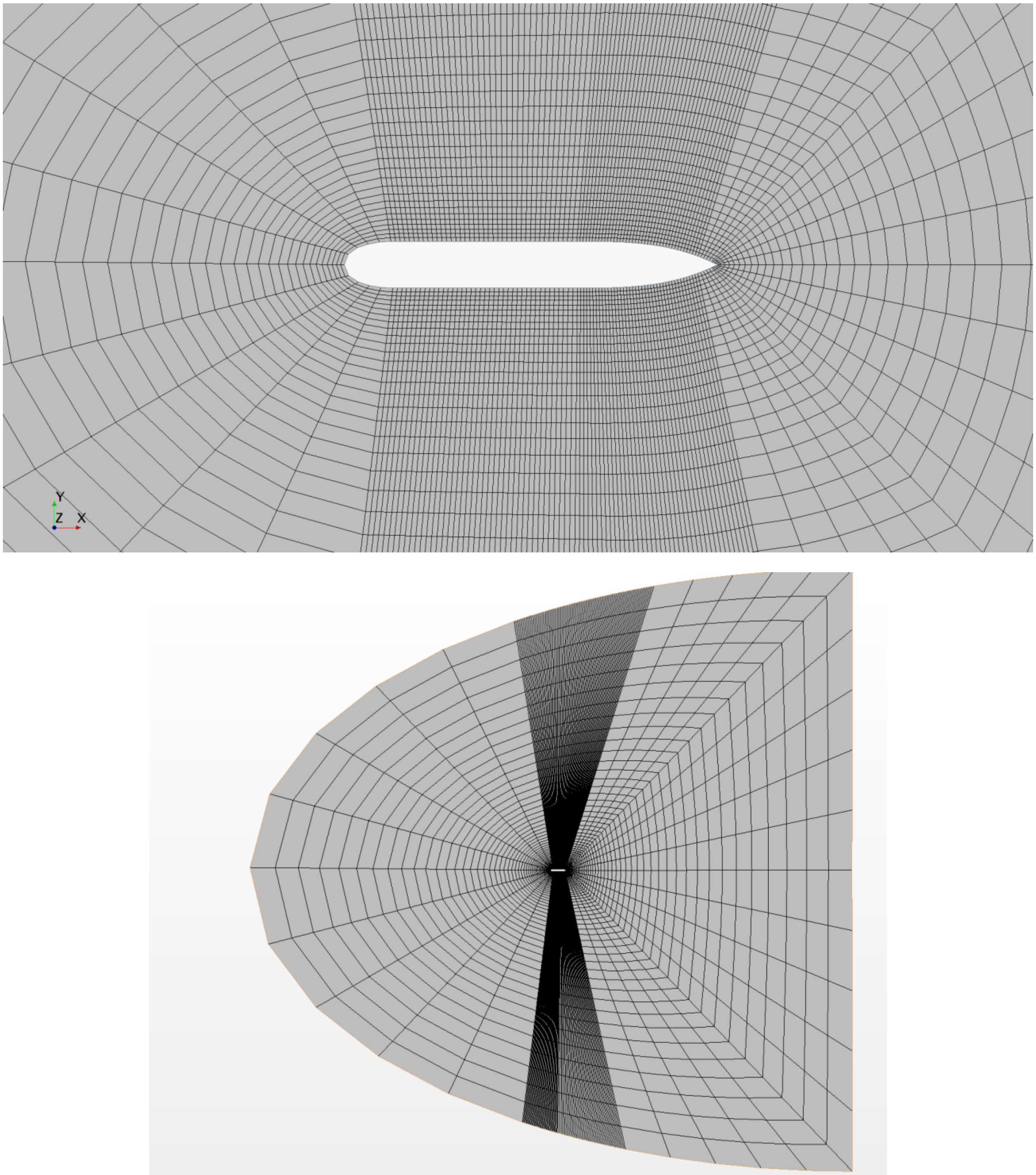


Figure 8. Locally dense directed mesh

3. Set the simulation physics

3a. The first physics is set to be inviscid, coupled energy, coupled flow and **steady**. We call it steady solver.

3b. The second physics is set to be inviscid, coupled energy, coupled flow and **explicit unsteady**. We call it unsteady solver

4. Run the simulation

A general strategy to yield non-unique solution is to run the simulation at a non-zero angle of attack (AOA) for enough iterations until the Mach contour evolves to asymmetric. Then switch AOA to zero until the Mach contour stays unchanged and the lift coefficient stays constant. Then, this stable solution is considered one of the non-unique solution at zero AOA. Let's assume we yield a positive lift at zero AOA, then we say ourselves are on the P-branch. Switch the AOA higher and lower to propagate on P-branch to know about the behavior at positive and negative AOAs on P-branch or jump from P-branch to other branches.

Simulations

1. Simulate on triangular mesh (Figure 6), steady solver

Initially, we set AOA=0.05, after iteration=1002, switch to AOA=0. When iteration=2000, we find the final Mach contour looks like Figure 9:

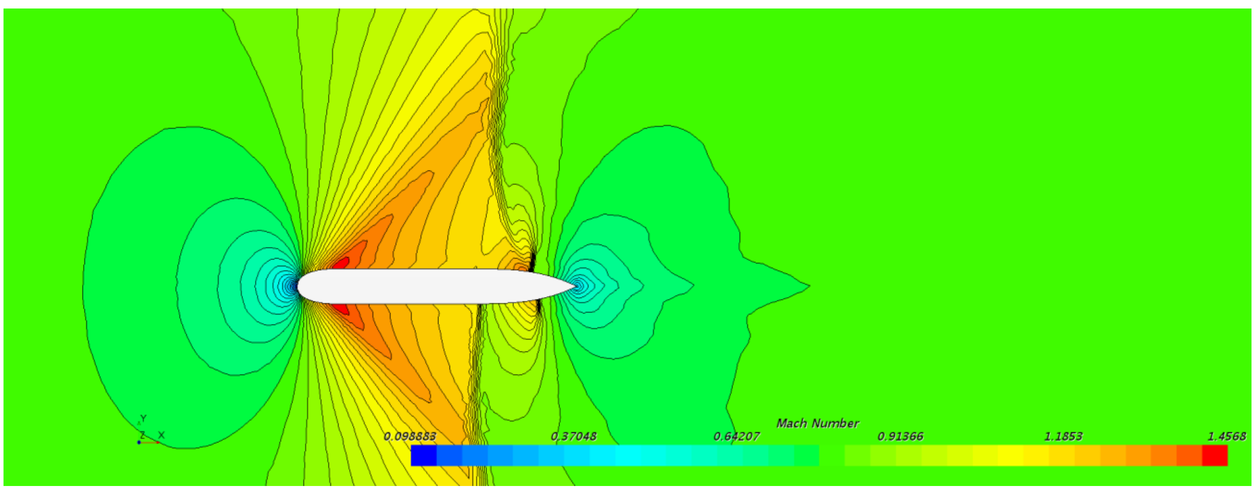


Figure 9. Mach contour for simulation 1

The residuals of Continuity, X-Momentum, Y-Momentum and Energy Equations are shown in Figure 10.

Note the sudden jump at iteration=1000 indicates the switch of AOA from 0.05 to 0.

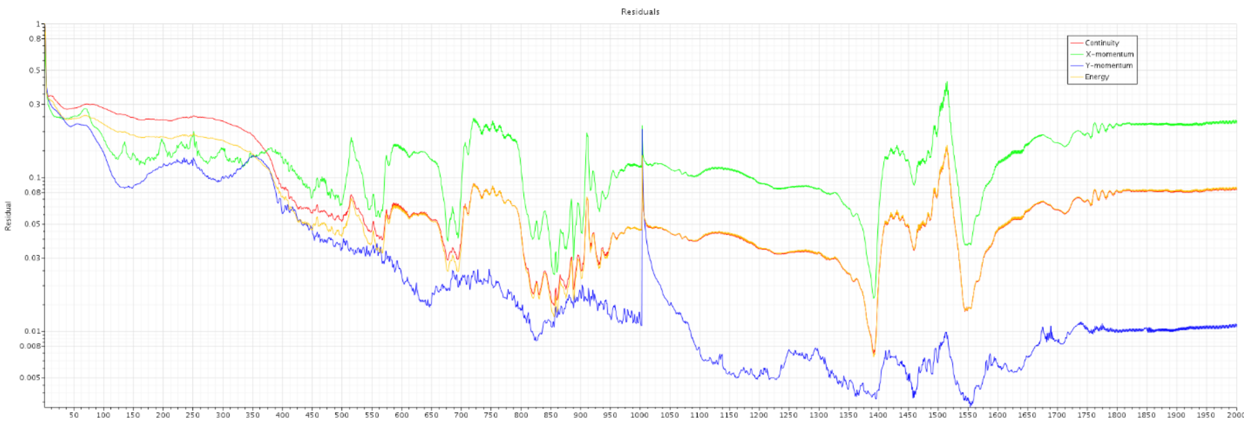


Figure 10. Residuals for simulation 1

The C_p-x plot is shown in Figure 11. Note the sudden jumps at x=0.65, 0.825 and 0.85 are caused by the shocks below and above the airfoil. It can be speculated by Figure 9.

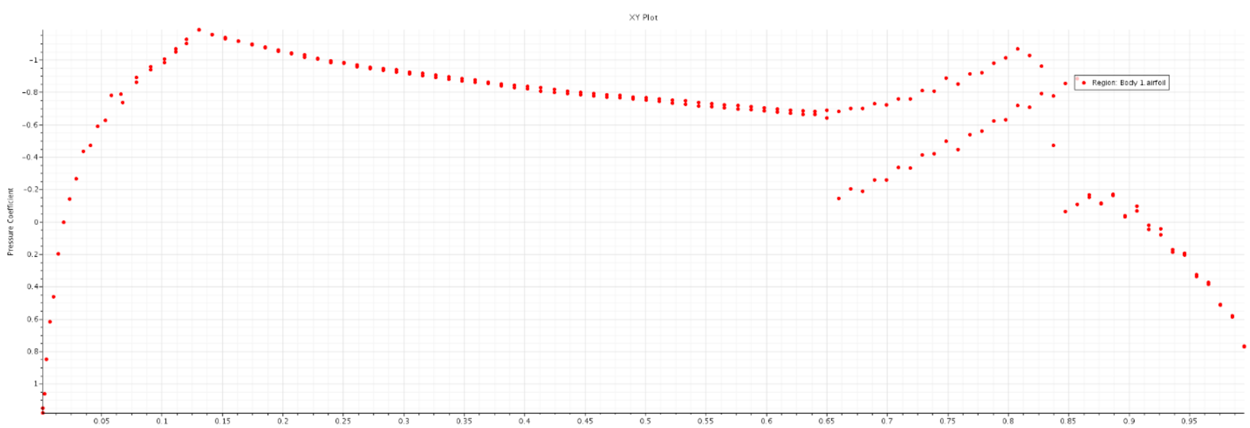


Figure 11. Cp-x plot for simulation1

The change of lift coefficient with respect to iterations is shown is Figure 12. Note the steadiness of lift coefficient is a sign of the stable of the solution. Note the zero AOA lift coefficient is found to be ~0.06, which is basically the same as the reference result (Figure 3).

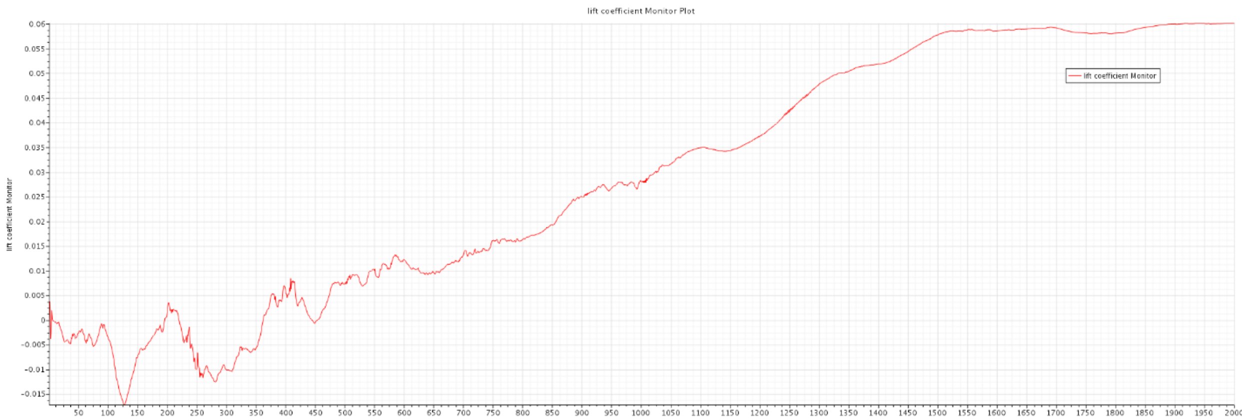
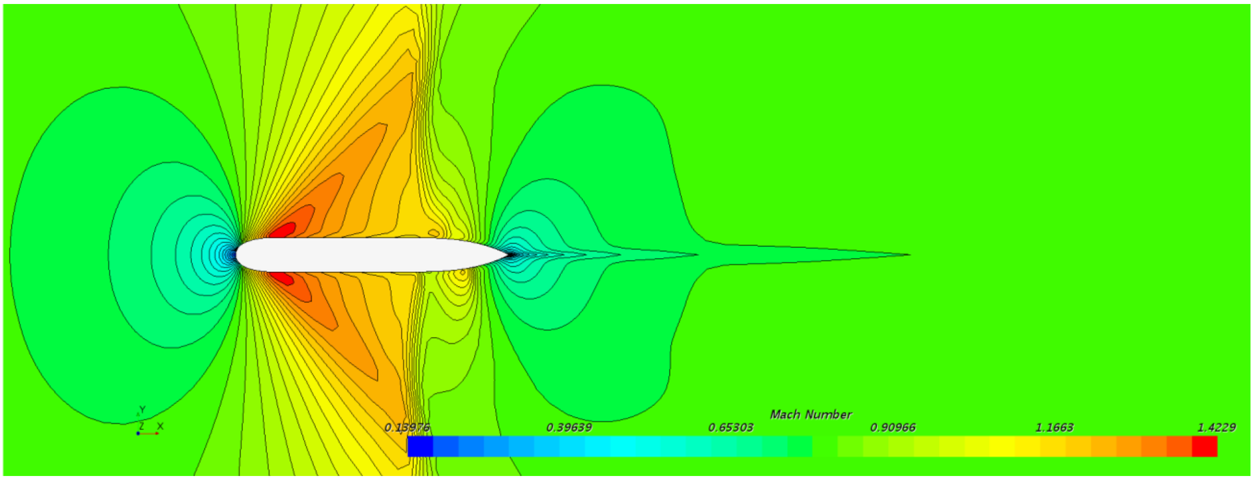


Figure 12. Lift coefficient plot for simulation 1

One shortcoming for triangular mesh is that there are wiggles along the shocks and Mach contours (Figure 9). To better simulate the shocks, directed meshes are preferred.

2. Simulate on coarse directed mesh (Figure 7), steady solver

As usual, first set AOA=0.05, iterate 803 steps, Mach contour is shown below:



Switch back to AOA=0, iterate until 2000 steps, then the Mach contour is shown in Figure 13. We see the shocks are straighter than those in Figure 9.

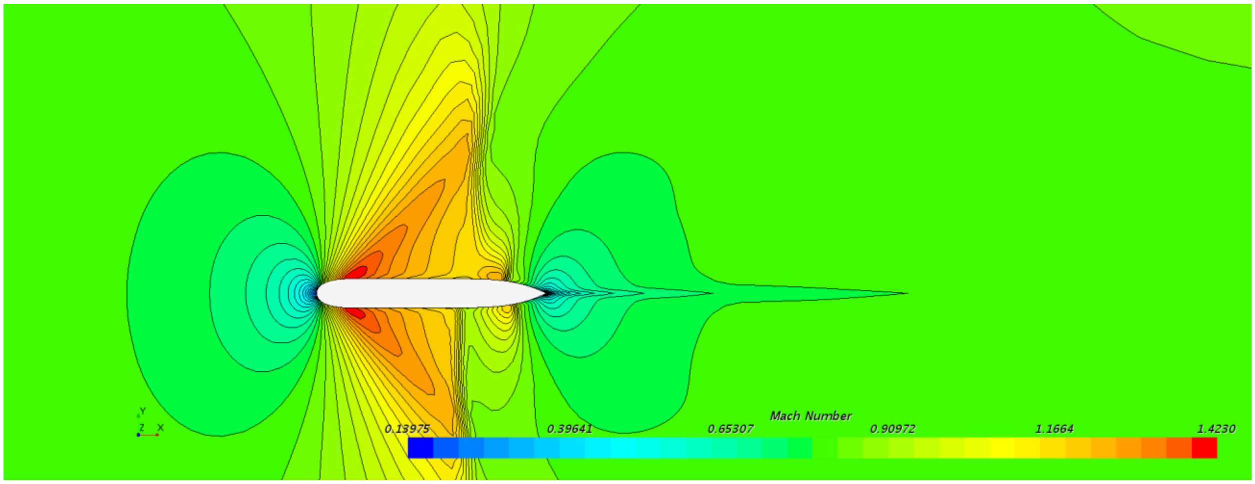


Figure 13. Mach contour for simulation 2.

The lift coefficient converges to ~0.0588, which matches the reference result better (Figure 3).

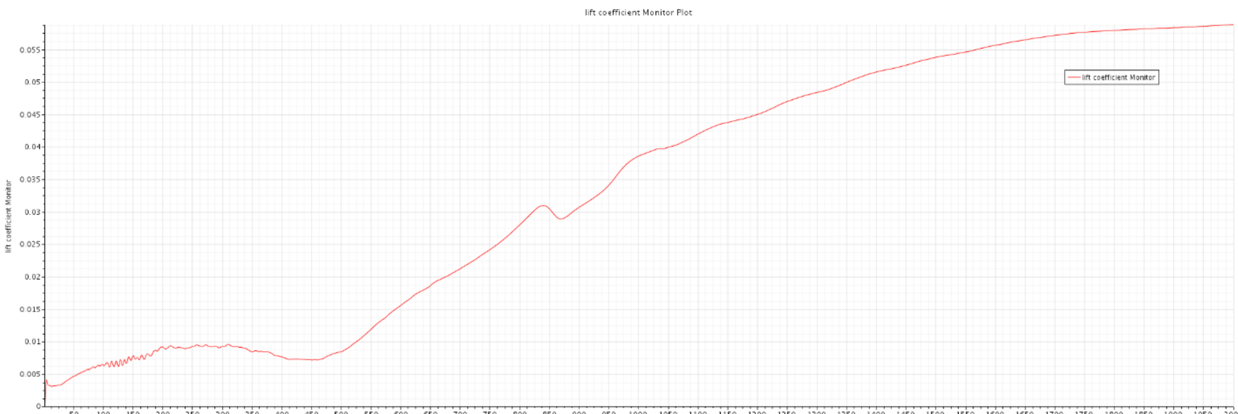
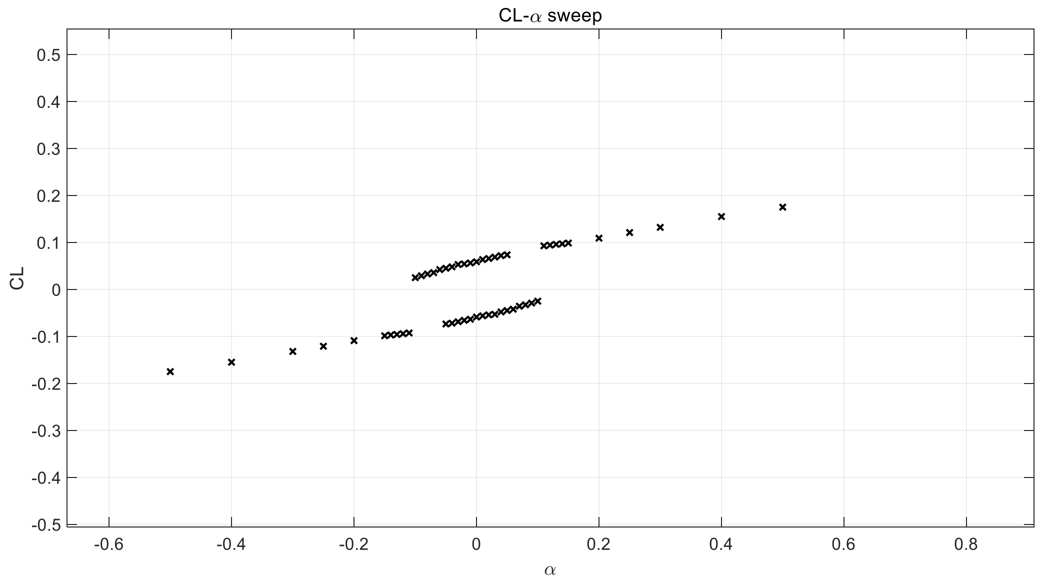


Figure 14. Lift coefficient for simulation 2

If we sweep α from 0 to 0.05 degrees and from 0 to -5 degrees, we get a C_L-α plot shown in Figure 15.

Note, different from result in Figure 3, Figure 15 shows the jump from P-Branch to N-Branch occurs at AOA=-0.1 deg instead of -0.05 deg. This indicates the actual lift coefficient sweep details can be interfered by mesh and resolution.



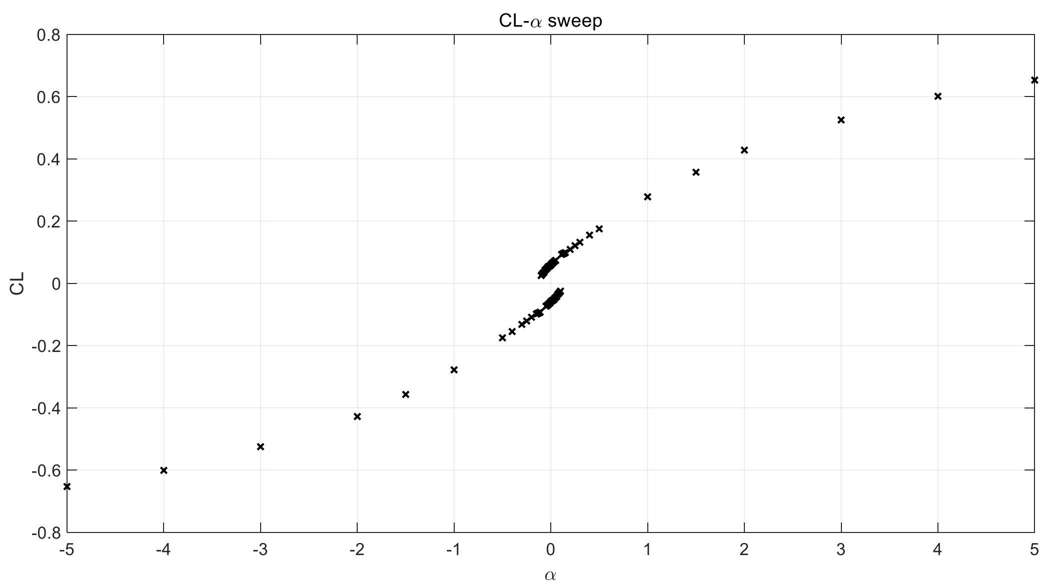


Figure 15. C_L-α sweep for simulation 2 (The above is a zoomed-in version)

To figure out if mesh effect really affects the detail of non-unique solution, a denser mesh is applied to simulate.

3. Dense directed mesh (400*150), steady solver

This time, AOA=0.15 and run for~2604 iterations. Switch to AOA=0. Note, for starting AOA<0.15, it is hard to get to P-Branch. We find the Mach contour looks like Figure 16. And the lift coefficient monitor Figure 17 gives a steady lift coefficient of ~0.0425. It is very different from reference result (Figure 3). This may be because the resolution of the mesh differs from the reference mesh very much. The reference mesh resolution is ~320*64, which is about only a third of our dense directed mesh (400*150). This discrepancy further validates the argument that mesh has effect on details of non-unique solutions, but different meshes all give the conclusion of the existence of non-unique solutions.

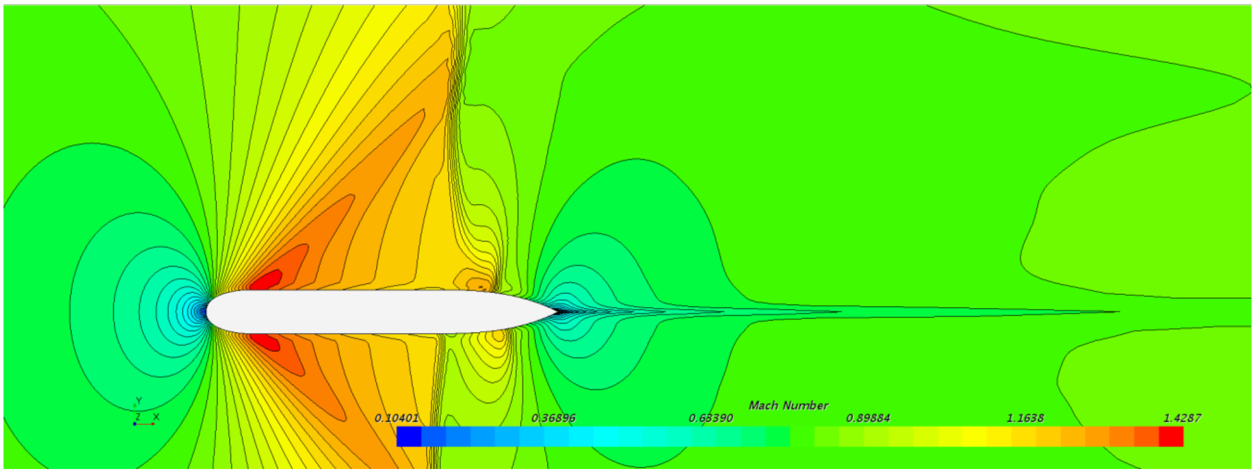


Figure 16. Mach contour for simulation 3

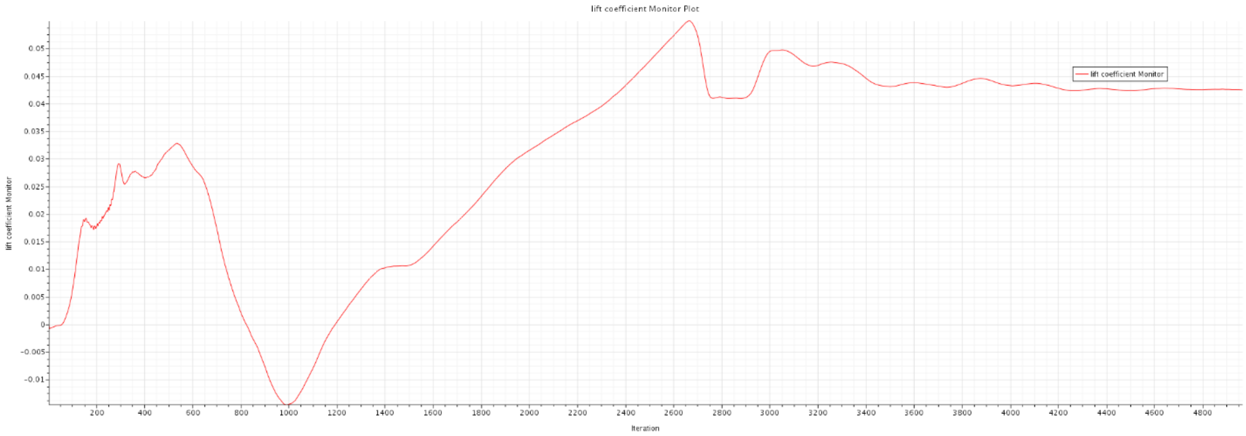
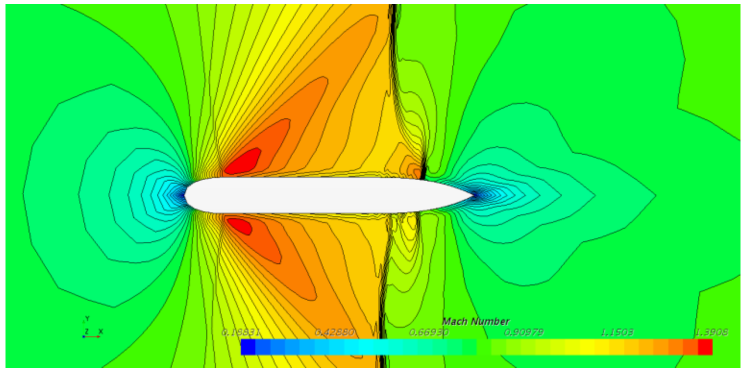
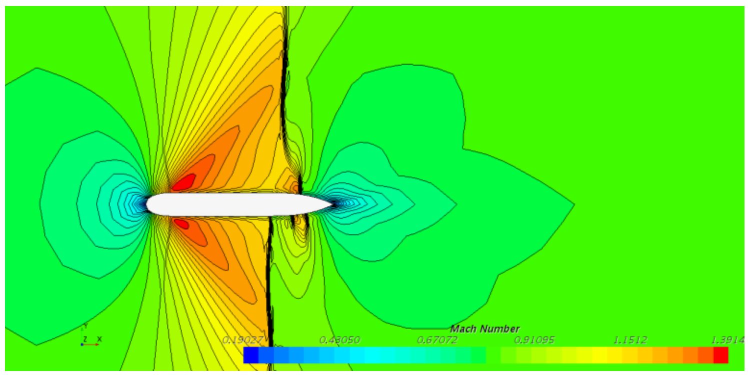


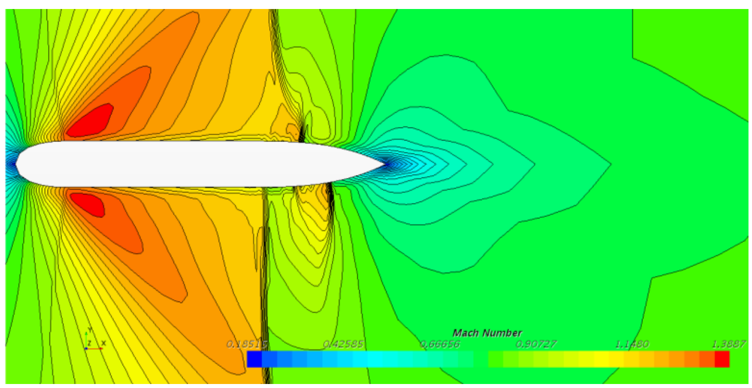
Figure 17. Lift coefficient monitor for simulation 3

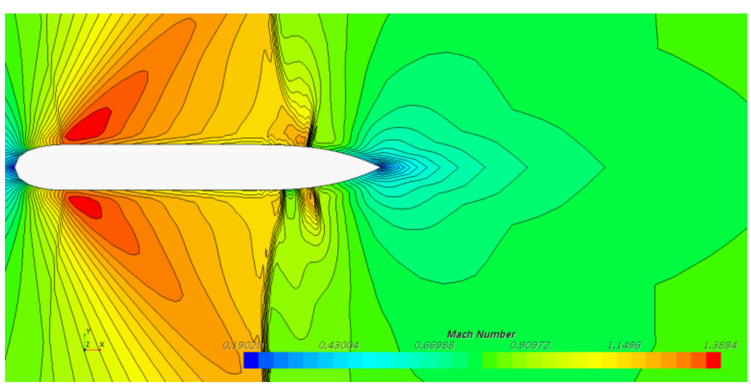
4. Locally dense mesh (Figure 8), steady solver

This time, to study if shocks can be simulated well by the very locally dense mesh, and further study mesh effect, we apply the same simulation on locally dense mesh. We find the simulation cannot be stabilized. The following are snapshots of the Mach contours during the simulation (Figure 18): We cannot determine whether the solution stays on P-Branch or N-Branch, instead, it jumps between branches. This shows some mesh even lead to unsteady non-unique solution. It is necessary to perform unsteady solver to the same problem. It can be argued that a denser mesh at shock places can surely improve the simulation of shock. (Figure 18) Note the very thin regions where the Mach contours concentrate.


Mach Number
0.18831 0.42880 0.66930 0.90979 1.1503 1.3908


Mach Number
0.19027 0.43050 0.67072 0.91095 1.1512 1.3914


Mach Number
0.18515 0.42585 0.66656 0.90727 1.1480 1.3887


Mach Number
0.19020 0.43004 0.66988 0.90972 1.1496 1.3894

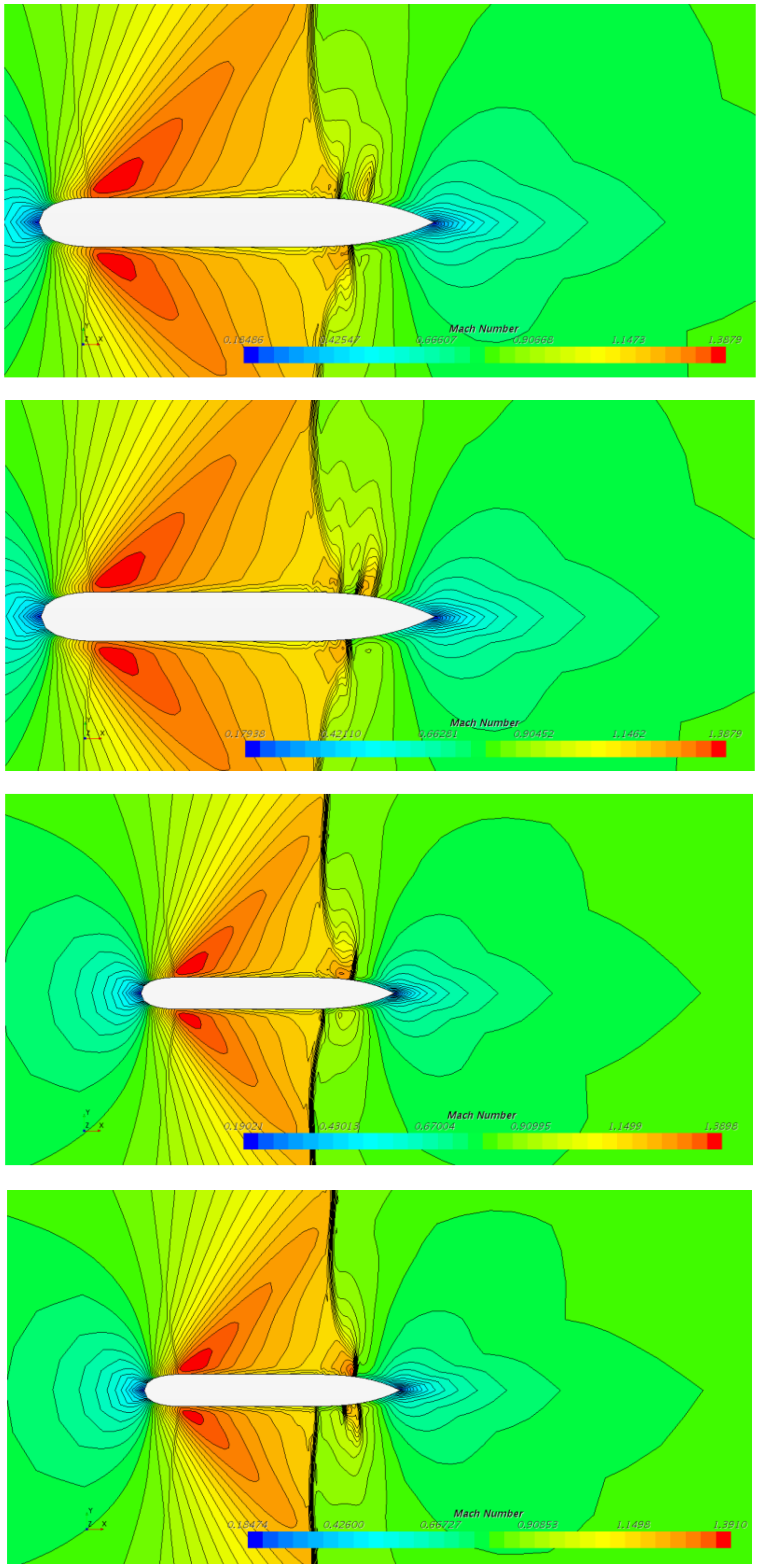
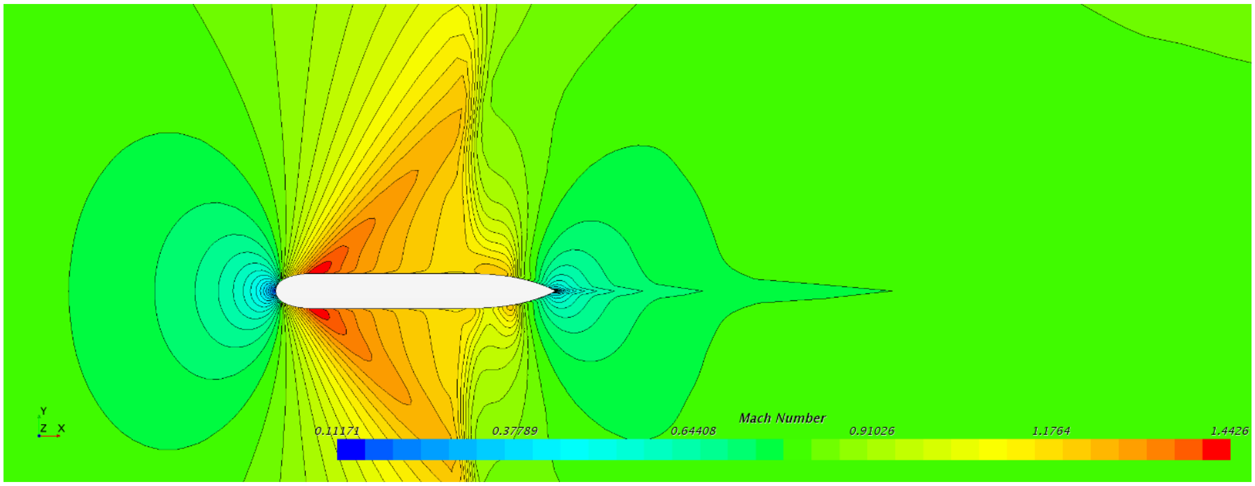


Figure 18. Mach contour evolution in sequence of iteration stepsfor simulation 4

5. Coarse directed mesh (Figure 7), unsteady solver

As usual, set AOA=0.05, we iterate 13040 steps, when t=5.046637e-2s, the Mach contour looks like:



Then switch to AOA=0. After 36241 iterations, when t=1.395255e-1s the Mach contour turns out to be symmetric (Figure 19). And this remains steady.

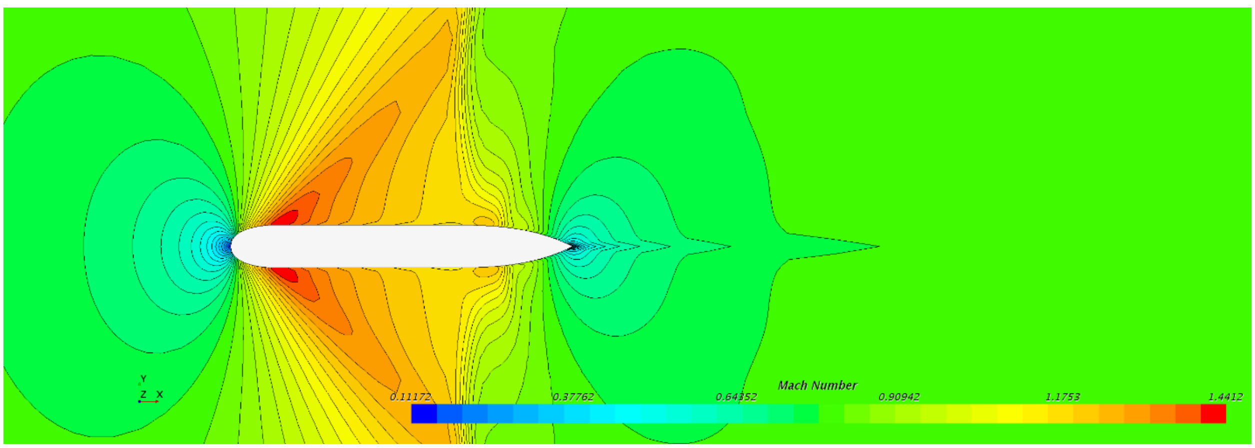


Figure 19. Mach contour for simulation 5

The result here shows the non-unique solution may disappear if unsteady solver is applied. The use of unsteady solver is more likely to reflect the true physical situation.

Conclusions and Discussion

1. Our study applies STAR-CCM+ solvers to generate non-unique solutions for airfoil JB1 on inviscid and transonic conditions. Simulations compare the non-unique solutions given by:

a) 2 different types of meshes (triangular and directed)

b) 2 directed meshes of different resolutions

c) 2 different types of time regime (steady and explicit unsteady)

2. Simulation results show:

a) Different types of meshes of similar resolution can give rise to different non-unique solution details, such as shock shape, lift coefficient and lift coefficient jump locations on C_L-α curves.

b) Different resolutions can give rise to large difference in lift coefficient result and shock locations.

c) Special mesh shapes can sometimes cause the steady solver to generate unsteady solution.

d) Unsteady solver may sometimes be better than the steady solver to obtain a more physical result.

3. Actual experiments needs to be carried out to show whether the lifting solution at zero AOA physically exists. If it is proved that this phenomenon is caused by the nature of governing equations, more sophisticated algorithms need to be developed to deal with non-unique solutions, either proves it to be physical in certain conditions or proves it to be trivial to be neglected.